**BACHELOR THESIS – ME 141502**

# FLUID FLOW ANALYSIS OF JACKET COOLING SYSTEM FOR MARINE DIESEL ENGINE 93 KW

Fadhil Rizki Clausthaldi
NRP 4212 101 023

Supervisor
Ir. Aguk Zuhdi Muhammad Fathallah, M.Eng, Ph.D
Dr.-Ing Wolfgang Busse

DEPARTMENT OF MARINE ENGINNERING
Double Degree Marine Engineering (DDME) Program
Faculty of Marine Technology
Hochschule Wismar, University of Applied Science
Institut Teknologi Sepuluh Nopember
Surabaya
2017

*“This Page Intentionally Left Blank*

# APPROVAL FORM

**Fluid Flow analysis of Jacket Cooling System for Marine Diesel Engine 93 Kw**

**Bachelor Thesis**

Submitted to Comply the Requirements to Obtain a Bachelor Engineering Degree
On
Double Degree Marine Engineering (DDME) Program
Department of Marine Engineering
Faculty of Marine Technology
Institute Teknologi Sepuluh Nopember
Hochshule Wismar, University of Applied Sciences

Submitted by
**FADHIL RIZKI CLAUSTHALDI**
**NRP. 4212 101 023**

Approved by Supervisor of Bachelor Thesis:

Dr. Ir. Aguk Zuhdi M. F., M.Eng.
NIP. 1956 0519 1986 10 1001

Dr. Ing. Wolfgang Busse

SURABAYA
January, 2017

*"This Page Intentionally Left Blank"*

# APPROVAL FORM

**Fluid Flow analysis of Jacket Cooling System for Marine Diesel Engine 93 Kw**

**Bachelor Thesis**

Submitted to Comply the Requirements to Obtain a Bachelor Engineering Degree
On
Double Degree Marine Engineering (DDME) Program
Department of Marine Engineering
Faculty of Marine Technology
Institute Teknologi Sepuluh Nopember
Hochshule Wismar, University of Applied Sciences

Submitted by
**FADHIL RIZKI CLAUSTHALDI**
**NRP. 4212 101 023**

Approved by Head of Marine Engineering Department

KEMENTERIAN RISET TEKNOLOGI DAN PENDIDIKAN TINGGI
INSTITUT TEKNOLOGI SEPULUH NOPEMBER
FAKULTAS TEKNOLOGI KELAUTAN
JURUSAN TEKNIK SISTEM PERKAPALAN

Dr.Eng. Muhammad Badrus Zaman, ST. MT.
NIP. 1977 0802 2008 01 1007

**SURABAYA**
January, 2017

*"This Page Intentionally Left Blank"*

# APPROVAL FORM

**Fluid Flow analysis of Jacket Cooling System for Marine Diesel Engine 93 Kw**

**Bachelor Thesis**

Submitted to Comply the Requirements to Obtain a Bachelor Engineering Degree
On
Double Degree Marine Engineering (DDME) Program
Department of Marine Engineering
Faculty of Marine Technology
Institute Teknologi Sepuluh Nopember
Hochshule Wismar, University of Applied Sciences

Submitted by
**FADHIL RIZKI CLAUSTHALDI**
**NRP. 4212 101 023**

Approved by Head Representative of Hochschule Wismar in Indonesia

for

Dr.-Ing. Wolfgang Busse

*"This Page Intentionally Left Blank"*

# DECLARATION OF HONOR

I, who signed below hereby confirm that:

*This final project report has written without any plagiarism act, and confirm consciously that all the dates, concepts, design, references, and material in this report own by Marine Power Plant Laboratory (MPP) in Department of Marine Engineering ITS which are the product of research study and reserve the right to use for further research study and its development.*

| | |
|---|---|
| Name | : Fadhil Rizki Clausthaldi |
| NRP | : 4212 101 023 |
| Bachelor Thesis Title | : ***Fluid Flow Analysis of Jacket Cooling System for Marine Diesel Engine 93 Kw*** |
| Department | : Marine Engineering |

If there is any plagiarism in the future, I will fully responsible and receive the penalty given by ITS according to the regulation applied.

Surabaya, 05 January 2017

Fadhil Rizki Clausthaldi

*"This Page Intentionally Left Blank"*

# ABSTRACT

**Name** : **Fadhil Rizki Clausthaldi**
**NRP** : **4212101023**
**Department** : **Marine Engineering**
**Supervisors** : **1. Ir. Aguk Zuhdi M. Fathallah, M.Eng, Ph.D**
**2. Dr. Ing. Wolfgang Busse**

The main function of cooling system is to control the temperature in the engine. To know the flow in the jacket cooling system for marine diesel engine 93 KW and the couple simulation between 1D modeling and 3D modeling, the methodology used. The analysis process was performed by using 3 diffrent softwares. The methodology to analysis fluid flow is CFD (computational fluid dynamic) with steps were problem identification, literature study, design the jacket cooling system based on the cummin diesel engine 93 KW, 1D modeling of cooling system, 3D modeling fluid flow in jacket cooling system, and conclusion. The input of 3D jacket cooling simulation are mass flow, fluid temperature, wall temperature, and heat transfer. The result from this bachelor thesis is fluid flow in jacket cooling system and another parameter output such as temperature flow and velocity if fluid in the jacket cooling system. The result of the flow in jacket cooling is much turbulance in various are of jacket cooling its mean the jacket cooling have a good enfficiency of heat transfer, and the fluid temperature show the increasing temperature from inlet to outlet because of heat transfer happen in the jacket cooling between wall of jacket cooling and fluid. The engine speed will affect the cooling system, if the engine speed is increasing, the speed of flow will increase because the cylinder block need more coolant and the temperature of cylinder block will increase

***Key Word*** *: Jacket cooling system, fluid flow, simulation, 3D modeling*

# ABSTRAK

**Nama** : **Fadhil Rizki Clausthaldi**
**NRP** : **4212101023**
**Jurusan** : **Teknik Sistem Perkapalan**
**Pembimbing** : **1. Ir. Aguk Zuhdi M. Fathallah, M.Eng, Ph.D**
**2. Dr. Ing. Wolfgang Busse**

Fungsi utama dari sistem pendingin adalah untuk mengontrol suhu di mesin. Untuk mengetahui aliran dalam sistem pendingin jaket untuk mesin diesel laut 93 KW dan simulasi pasangan antara pemodelan 1D dan 3D, metodologi yang digunakan. Proses analisis dilakukan dengan menggunakan 3 software yang berbeda. Metodologi untuk aliran fluida analisis CFD (computational fluid dynamic) dengan langkah-langkah yang identifikasi masalah, studi literatur, merancang sistem jaket pendingin berdasarkan jintan diesel engine 93 KW, 1D pemodelan sistem pendingin, 3D aliran fluida pemodelan dalam sistem pendingin jaket , dan kesimpulan. Masukan dari 3D simulasi jaket pendingin adalah aliran massa, temperatur fluida, temperatur dinding, dan transfer panas. Hasil dari tugas akhir sarjana ini adalah aliran fluida dalam sistem pendingin jaket dan output parameter lain seperti aliran suhu dan kecepatan jika cairan dalam sistem jaket pendingin. Hasil aliran pendinginan jaket banyak turbulance dalam berbagai area pada jaket pendinginan yang berarti pendinginan jaket memiliki enfficiency baik terhadap perpindahan panas, dan suhu fluida menunjukkan suhu meningkat dari inlet ke outlet karena perpindahan panas terjadi di jaket pendinginan antara dinding pendingin jaket dan cairan. Kecepatan mesin akan mempengaruhi sistem pendingin, jika kecepatan mesin meningkat, kecepatan aliran akan meningkat karena blok silinder membutuhkan lebih banyak cairan pendingin dan suhu blok silinder akan meningkat.

***Kata Kunci*** *: sistem pendingin jaket, aliran fluida, simulasi, model 3D, CFD*

# PREFACE

Bismillahirrahmaanirrohiim

Alhamdulillahirabbilalamin, by the blessing of Allah SWT. The author is given strength and ease to accomplish the paper script in time. This paper, by title "**Fluid Flow analysis of Jacket Cooling System for Marine Diesel Engine 93 Kw**"is submitted in fulfillment for Double Bachelor's Degree Program in Institut Teknologi of Sepuluh Nopember and Hochschule Wismar.

The author realizes that this task was not done by one man's efforts, but it was also supported by all sorts of sides surrounding the author. Therefore, the author would say thanks' to:

1. My beloved family, my mother Meta Mahendradatta for incomparable love, patience and prayers. My Father Abu Bakar Tawali for spirit and motivation of the author in completing this thesis.
2. My Brothers and my sister, for their love, supports and happiness in the family.
3. Ir.Dwi Priyanta MSE as the lecture guidance author in Marine Engineering FTK ITS.
4. Dr.Eng.M.Badrus Zaman,S.T., M.T as the head of Marine Engineering department Institut teknologi sepuluh nopember.
5. Ir. Aguk Zuhdi M. Fathallah, M.Eng, Ph.D as the supervisor 1 who has guided the author and motivation to accomplish the thesis and gave a suggestion all the time

6. Dr. Ing Wolfgang Busse as the supervisor 2 who has guided the author and motivation to accomplish the thesis.
7. Dr. Ir. A.A. Masroeri who has guided the author and motivation to accomplish the thesis.
8. Lecture’s In Marine Engineering department who has given beneficial knowledge.
9. Friends in the Laboratory of Marine Power Plant and MMD who contribute in the completion of the thesis.
10. Akbar alrasjid and Aji suryadi who help me and teach me about the software to accomplish the thesis
11. Friends 2012 Bismarck forces who have provided support and motivation.

Finally, the authors realize that this bachelor thesis is still not perfect yet. Therefore the authors expect some recommendation from the readers to make it better.

Surabaya, January 2017

Author

# TABLE OF CONTENT

# TABLE OF FIGURES

# LIST OF TABLES

*"This Page Intentionally Left Blank"*

# CHAPTER 1
# INTRODUCTION

## 1.1 Background

Developments of science and technology are very quick to give a good impact as well as a great benefit for humans in many areas of life. It can be seen by the increasing number of equipment that has been created by humans in a variety of models, shapes and capabilities in terms of the use of relatively superior to conventional equipment. (Wibowo, 2006).

Marine diesel engine is widely used for the needs in the field of marine propulsion, because the shape is compact and economic. For marine diesel engine with 93 kW power are needed for vessels with a tonnage between 20-30 GT (especially for fishing vessels). Associated with one of the research laboratories in motor fuel, Shipping Systems Engineering Department, Faculty of Marine Technology, Institute of Technology will require an analysis related to the performance of marine diesel that had been established earlier by the method of reverse engineering.

In many engines, one of the most important system in engine is cooling system. Cooling system has a purpose to cooling down the part of the engine. The part of engine hes a different work which made the part has a high temperature, the temperature of the component of the engine has a limits, if the temperature through the limits, it will be dangerous for the engine and can made the engine breakdown and cannot work anymore. So in this case, the cooling system is to maintain the temperature of the engine to make the temperature steady in the normal temperature. The fluid of the cooling system has a standard temperature to cooling down the engine components.

Juniono Raharjo, Institut Teknologi Sepuluh Nopember (ITS) student's bachelor thesis in 2015, it discusses about "Design marine diesel using Reverse Engineering Method" fishing vessel with capacity around 30. The result of the thesis is engine design with 93 KW. In this engine. There are some parts that have not been include jacket cooling system.

## 1.2 Statement of Problem

Based on the above description, presented several formulation of the problem

1. How to simulate the jacket cooling system for marine diesel engine 93 Kw using CFD method?
2. How the flow of jacket cooling system marine diesel engine 93 KW?

## 1.3 Research Limitation

1. Analysis of the research using software
2. The components of cooling system in marine diesel engine 93KW
3. Using computational fluid dynamics and coupling between 1d and 3D models

## 1.4 Research Objectives

This bachelor thesis aim to:

1. Knowing the simulation for jacket cooling system for marine diesel engine 93kw using CFD method
2. Knowing the result of the design of jacket cooling system in marine diesel engine that is good of bad condition

## 1.5 Research Benefits

Benefits to be gained from this bachelor thesis are:

1. Reference in utilization of fluid analysis cooling system

*“This Page Intentionally Left Blank”*

# CHAPTER 2
# STUDY LITERATURE

## 2.1 State of the art

### 2.1.1 Computational Fluid Dynamic Analysis

Fluid flows are governed by partial differential equations which represent conservation laws for the mass, momentum, and energy. Computational Fluid Dynamics (CFD) is the art of replacing such PDE systems by a set of algebraic equations which can be solved using digital computers.

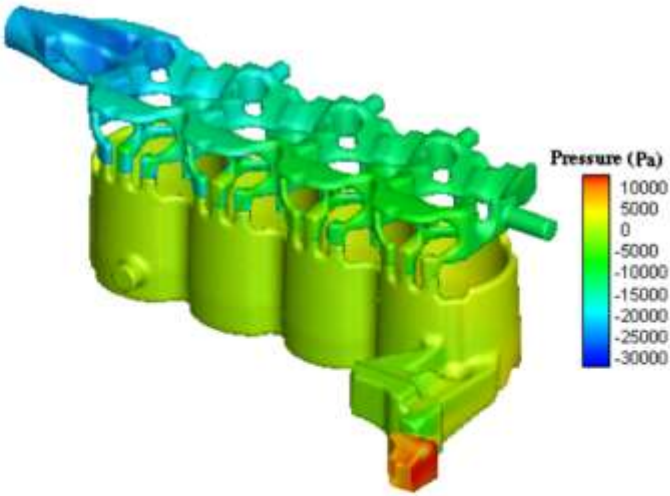



Figure 2.1 Total pressure distribution in coolant jacket
Source : A. Mohammadi and M. Yaghoubi (2011)

CFD has made rapid advances over the years and is now used as an effective tool in the analysis and visualization of fluid flows in complex systems including the engine cooling jackets. Along with visualizing the flow development in the jacket passages, CFD techniques are also applied to estimate temperature distribution over the entire engine block. It also helps to study and understand complex phenomena that commonly occur in cooling jackets, like cavitations and nucleate boiling.

Juniono Raharjo, Institut Teknologi Sepuluh Nopember (ITS) student's bachelor thesis in 2015, it discusses about "Design marine diesel using Reverse Engineering Method" because fishing vessel with capacity around 30 GT has problem with low availability of the ship engine, which most of the ships still use non marine diesel engine as its main propulsion. The result of the thesis is engine design with 93 KW. There are many analysis has ben done in this marine diesel engine such as, performance prediction, motion analysis, stress analysis, turbocharger.

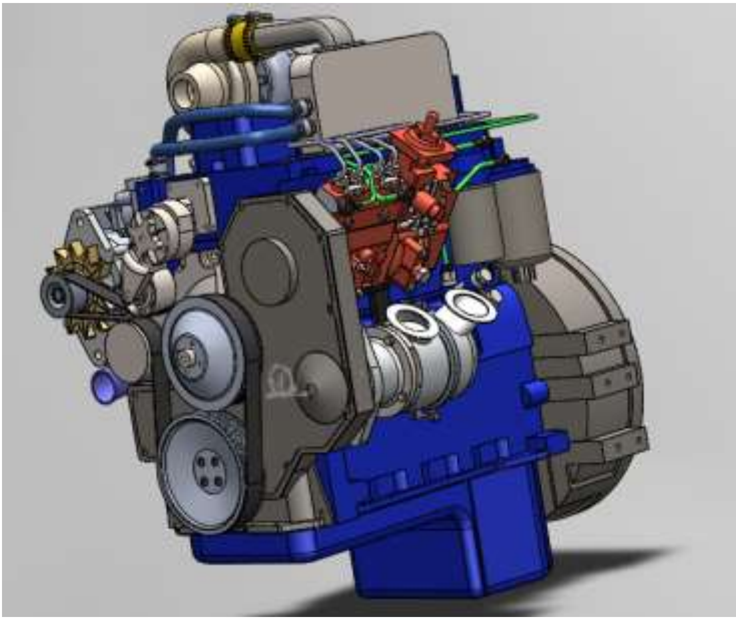


Figure 2.2 Design of Marine Diesel Engine 93Kw
Source : Juniono Raharjo (2010)

The complex shape of the cooling jacket is influenced by multiple factors including the shape of the engine block and optimal temperature at which the engine runs. A very large cooling jacket would be effective in transporting heat away from the cylinders, however, too large of a geometry results in extra weight to be transported. Also, engineers would like the engine to reach its optimal operating temperature quickly. In the following, we describe the major components of the geometry and the design goals of the mechanical engineers responsible for the analysis. (Robert S. Laramee)

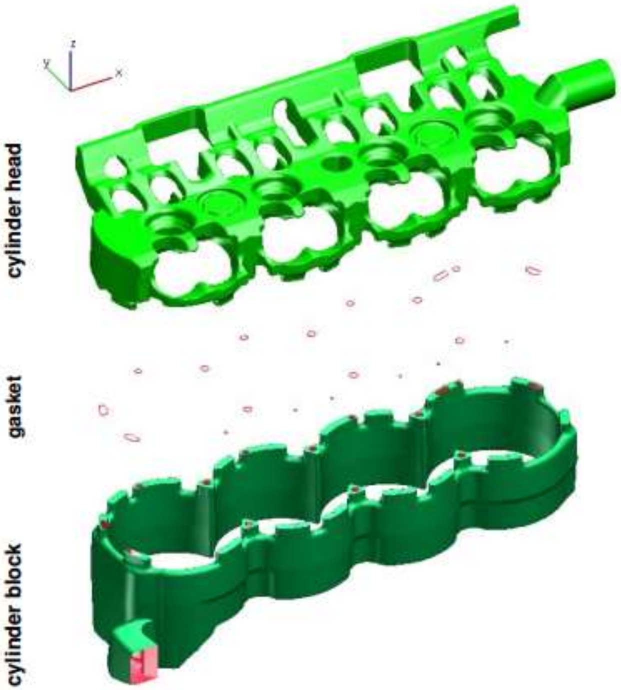



Figure 2.3 The cooling jacket has been split apart for illustration. The geometry consists of three primary components: (top) the cylinder head, (middle) the gasket, and (bottom) the cylinder block
Source : Robert S. Laramee (2010)

Even for a simple 1D cooling system model a lot data for different components needs to be obtained. It can once again be recommended for engineers to go through the GTISE tutorials (Gamma Technologies (2009) and try building simple models according it to become familiar with GT-ISE interface. In addition one should look at cooling systems and thermal management tutorials

(Gamma Technologies (2009) to get a better understanding on how the cooling model is supposed to operate and what are the recommended settings and parameters for template objects and the simulation itself. (Alexey Vdovin,2010)

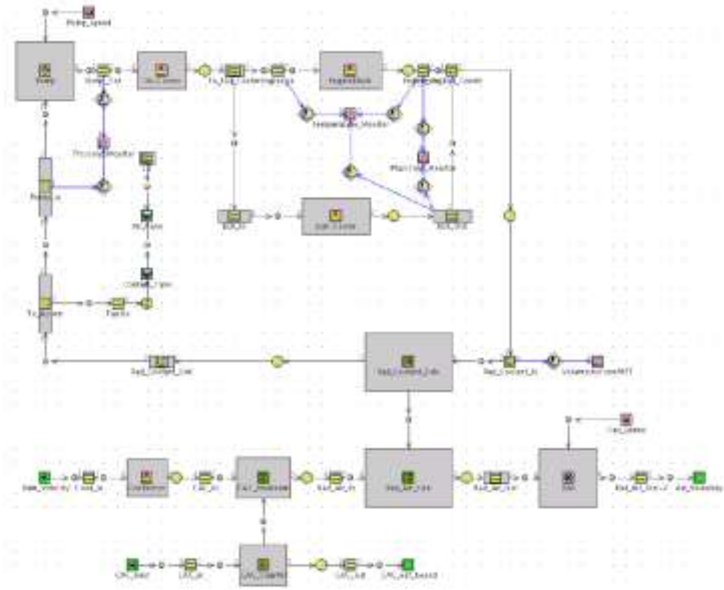


Figure 2.4 Model of a cooling system
Source : Alexey Vdovin (2010)

### 2.1.2 Coupling Methodology

In the introduction, it was established the need for the development of a coupling methodology in order to perform a simulation combining a 1D and a 3D code. The main differences between both codes have been described, including the difficulties to modify the commercial 3D software, FLUENT. Therefore, the chosen approach must allow an implementation in terms of the available tools of FLUENT (i.e. UDFs and monitors), without overlooking the accuracy in managing the transient problems that have been scoped. Eventually, the well-known Method of Characteristics was found to meet these requirements, and so was used as a means of solving the coupling problem. Some Refs. performence the coupling directly transferring the flow field variables, or by means of a Riemann solver. In contrast to those, the use of the MoC is a physically based methodology that will give the exact solution if the hypotheses are satisfied. Moreover, the implementation of boundary conditions by means of the MoC for different elements, such as volumes, has been the subject of a wide analysis in the past years and Open WAM has already a set of them implemented (J. Galindo, 2010)

## 2.2 Theoritical Beackground

### 2.2.1 Cooling System

The cooling system is a system of parts and fluid that work together to control an engine's operating temperature for optimal performance. The system is made up of passages inside the engine block and heads, a water pump and drive belt to circulate the coolant, a thermostat to control the temperature of the coolant, a radiator to cool the coolant, a radiator cap to control the pressure in the system, and hoses to transfer the coolant from the engine to the radiator

The main purpose of the cooling system is to remove the heat from the engine generated in the combustion process. In addition, the cooling system maintains the charge air and lubricating oil temperatures at nominal values. A secondary function for the system is to preheat the engine block during operating breaks. Indirect cooling meaning that the heat is released to a closed coolant circuit instead of releasing the heat directly to cooling air without using a cooling agent. In the cooling system the cooling water is circulated through the heat sources of the engine. The heat is conducted to the cooling water and the circulating motion carries the heat away from the heat sources. The cooling system also includes a method to remove the heat load from the operating cycle to the environment. For example radiators, cooling towers, central coolers and box coolers are used for this purpose. In most cases the cooling system consists of two separate cycles that are referred to as primary cycle and secondary cycle. The primary cycle is always a closed cycle that circulates the cooling water in the heat sources and is cooled down by a heat exchanger. Treated fresh water is used as a cooling agent on primary cycle and glycol is used as an additive to prevent freezing when operating in cold conditions. The heat exchanger in the primary cycle serves as a connection for the heat to transfer to the secondary cycle which then takes care of the heat disposal. There are also applications where the heat exchanger in the primary cycle serves directly as a heat exposal method example when using radiator

In applications the cooling system configurations used are water/water cooling with a semi-open secondary cycle, water/water cooling with an open secondary cycle or air/water cooling where the secondary cycle is cooled down by air flow. The cooling system with cooling towers is an example of water/water cooling with a semi-open secondary cycle. This configuration is used in power plant applications and the cooling effect is based on evaporation. The cooling system with a central cooler using sea water in the secondary cycle is an example of a water/water cooling with an open secondary cycle. This configuration is the most common cooling system used on marine applications

### 2.2.2 Thermodynamics

The First Law of Thermodynamics states:

**"Energy can neither be created nor destroyed, only altered in form."**

For any system, energy transfer is associated with mass and energy crossing the control boundary, external work and/or heat crossing the boundary, and the change of stored energy within the control volume. The mass flow of fluid is associated with the kinetic, potential, internal, and "flow" energies that affect the overall energy balance of the system. The exchange of external work and/or heat complete the energy balance.

The First Law of Thermodynamics is referred to as the Conservation of Energy principle, meaning that energy can neither be created nor destroyed, but rather transformed into various forms as the fluid within the control volume is being studied. The energy balance spoken of here is maintained within the system being studied. The system is a region in space (control volume) through which the fluid passes. The various energies associated with the fluid are then observed as they cross the boundaries of the system and the balance is made.

As discussed in previous chapters of this module, a system may be one of three types: isolated, closed, or open. The open system, the most general of the three, indicates that mass, heat, and external work are allowed to cross the control boundary. The balance is expressed in words as: all energies into the system are equal to all energies leaving the system plus the change in storage of energies within the system. Recall that energy in thermodynamic systems is composed of kinetic energy (KE), potential energy (PE), internal energy (U), and flow energy (PL); as well as heat and work processes. (DOE fundamental handbook volume 1,1992)

### 2.2.3 Heat Transfer

Actually, there is a distinct difference between the two. Temperature is a measure of the amount of energy possessed by the molecules of a substance. It is a relative measure of how hot or cold a substance is and can be used to predict the direction of heat transfer. The symbol for temperature is T. The common scales for measuring temperature are the Fahrenheit, Rankine, Celsius, and Kelvin temperature scales

In the case of combined heat transfer, it is common practice to relate the total rate of heat transfer , the overall cross-sectional area for heat transfer (Ao), and the overall temperature ˙ Q difference (ΔTo) using the overall heat transfer coefficient (Uo). The overall heat transfer coefficient combines the heat transfer coefficient of the two heat exchanger fluids and the thermal conductivity of the heat exchanger tubes. Uo is specific to the heat exchanger and the fluids that are used in the heat exchanger.(DOE fundamental handbook volume 2,1992)

$$Q = U_o A_o \Delta T_0 \qquad (2\text{-}1)$$

### 2.2.4 Fluid Flow

Several properties of fluids were discussed in the Thermodynamics section of this text. These included temperature, pressure, mass, specific volume and density. Temperature was defined as the relative measure of how hot or cold a material is. It can be used to predict the direction that heat will be transferred. Pressure was defined as the force per unit area. Common units for pressure are pounds force per square inch (psi). Mass was defined as the quantity of matter contained in a body and is to be distinguished from weight, which is measured by the pull of gravity on a body. The specific volume of a substance is the volume per unit mass of the substance. Typical units are ft3/lbm. Density, on the other hand, is the mass of a substance per unit volume. Typical units are lbm/ft3. Density and specific volume are the inverse of one another. Both density and specific volume are dependent on the temperature and somewhat on the pressure of the fluid. As the temperature of the fluid increases, the density decreases and the specific volume increases. Since liquids are considered incompressible, an increase in pressure will result in no change in density or specific volume of the liquid. In actuality, liquids can be slightly compressed at high pressures, resulting in a slight increase in density and a slight decrease in specific volume of the liquid.(DOE fundamental handbook volume 3,1992)

#### 2.2.4.1 Laminer Flow

Laminar flow is also referred to as streamline or viscous flow. These terms are descriptive of the flow because, in laminar flow, (1) layers of water flowing over one another at different speeds with virtually no mixing between layers, (2) fluid particles move in definite and observable paths or streamlines, and (3) the flow is characteristic of viscous (thick) fluid or is one in which viscosity of the fluid plays a significant part.

#### 2.2.4.2 Turbulent Flow

Turbulent flow is characterized by the irregular movement of particles of the fluid. There is no definite frequency as there is in wave motion. The particles travel in irregular paths with no observable pattern and no definite layers.

### 2.2.5 3D Modeling Software

Simulation of the stress distribution using ANSYS software to help the research. ANSYS software publishes engineering analysis software across a range of disciplines including finite element analysis, structural analysis, computational fluid dynamics, explicit and implicit methods, and heat transfer. In this research, the engineering analysis that will be used and obtained data is a result of calculation that already calculate by ANSYS software.

(www.ansys.com)

### 2.2.6 1D Modeling Software

GT-POWER is used to predict engine performance quantities such as power, torque, airflow, volumetric efficiency, fuel consumption, turbocharger performance and matching, and pumping losses, to name just a few. Beyond basic performance predictions, GT-POWER includes physical models for extending the predictions to include cylinder and tailpipe-out emissions, intake and exhaust system acoustic characteristics (level and quality), in-cylinder and pipe/manifold structure temperature, measured cylinder pressure analysis, and control system modeling. Standard GT-POWER engine models are easily converted to real-time capable models for SiL or HiL simulations. These models may also be included in a full system level simulation within GT-SUITE to provide accurate and physically based engine boundary conditions to the rest of the vehicle.

(www.gtisoft.com)

### 2.2.7 Solidwork

SOLIDWORKS MBD (Model Based Definition) is a built-in drawingless manufacturing solution for SOLIDWORKS. It helps companies define, organize, and publish 3D Product Manufacturing Information (PMI) in industry standard file formats such as 3D PDF and Drawings. Unlike traditional 2D drawings, by guiding the manufacturing process directly in 3D, SOLIDWORKS MBD allows for streamlined production, lower cycle times, and reduced errors.

SOLIDWORKS MBD sets data such as product models, dimensions, geometric tolerances, surface finishes, welding symbols, bill of materials (BOM), callouts, tables, notes, Meta properties, and other annotations within the SOLIDWORKS 3D environment in 3D PMI. Because all the information needed to guide the operation is integrated with the 3D models, traditional 2D drawings are no longer needed. The interactive 3D PMI provided by SOLIDWORKS MBD serves many operational use cases, such as part and assembly engineering drawings, Request for Quote (RFQ), and Inspection Reports. It also helps multiple departments and stakeholders across the operation, such as design, procurement, fabrication, assembly, quality, sales, marketing, clients, and suppliers. Figure below is a marine diesel engine solid work design

*"This Page Intentionally Left Blank"*

# CHAPTER 3
# METHODOLOGY

The methodology is a description of the steps carried out in a study. Methodology in this thesis include all activities carried out to solve a problem or process of analysis and evaluation of the problems this thesis. In general, the methods used in problem solving is diesel engine simulation modeling.

## 3.1 Flow Diagram

Flowchart diagram activities of this research can be seen in figure 3.1.

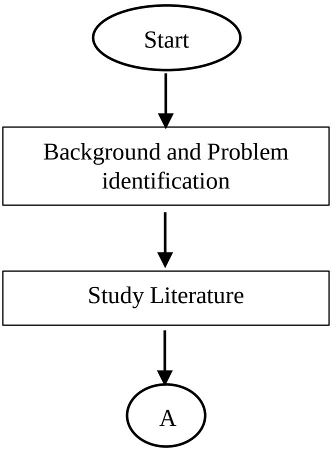



*Figure 3.1 Flowchart methodology research*

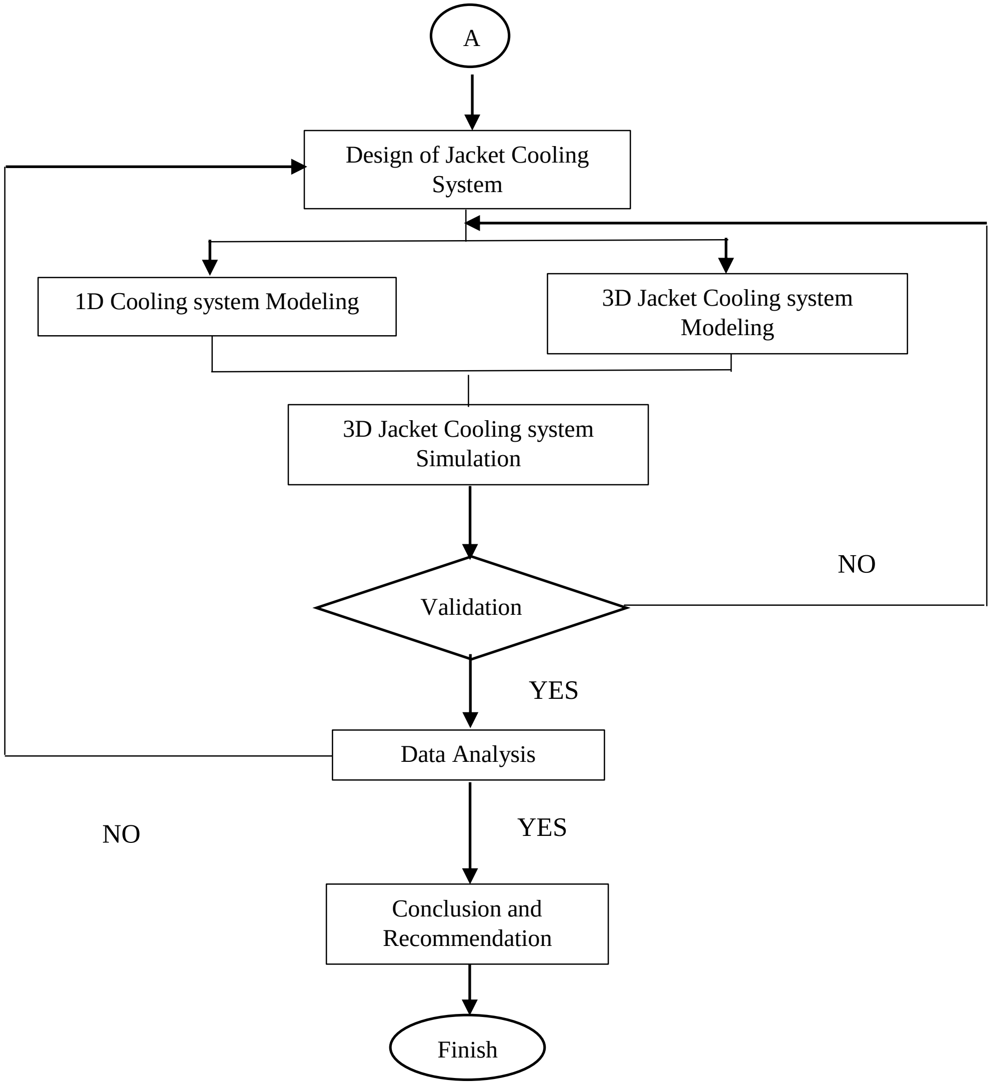



*Figure 3.2 Flowchart methodology research 2*

## 3.2 Background and Problem Identification

This thesis begins with identification and formulating the problems regarding to the experiment to be carried as well limitation issue. It aims to simplify the problem, make it compact and easier in the final project.

Explanation about the research background and Identification of the problem the research. The research is ti simulates the fluid flow cooling system.

## 3.3 Study Literature

The study of literature is the stage of learning about the basic theories to be discussed or used in this bachelor thesis. The basic theory of the cooling system, cooling system materials and about the software to simulates the flow of cooling system. The goal is to strengthen the basic theory problems as in the analysis from books, journal, and website. This stage also learning about design of the material which need to fulfill the requirement to do the simulation and analysis

## 3.4 Data Collection

After collecting the materials for the preparation of the basic theory, followed by collecting data such as engine component dimensions required for manufacturing simulation model. Data obtained from the design of marine diesel has been done.

Coolant Capacity - Engine Only……………………8.3litre

Maximum Engine Cooling Circuit External Resistance……………………………TBD

Minimum Pump Inlet Pressure with Open Thermostat and no Pressure Cap………………TBD

Maximum Static Head of Coolant Above Engine Crankshaft Centerline.………TBD

Standard (modulating) Thermostat Range…………82-93℃

Maximum Block Coolant Pressure with Closed Thermostat and no Pressure Cap………TBD

Minimum Pressure Cap……………………………… 50kPa

Maximum Engine Coolant Temperature at Engine Outlet……………………100℃

Maximum Engine Coolant Temperature for Engine Protection Devices……………101.6℃

Minimum Engine Coolant Temperature………………71℃

The data used in this bachelor thesis is the data of Marine Diesel Engine designed by Juniono Raharjo, student's of Marine Engineering Department – ITS Surabaya.

## 3.5 Design Of Cooling System

In this step is the creation of design for cooling jacket that will be used to make modeling in software. Design was done in software solidwork using dimensional data on an existing machine cumin. Dimension of the materials are taken manually in laboratorium

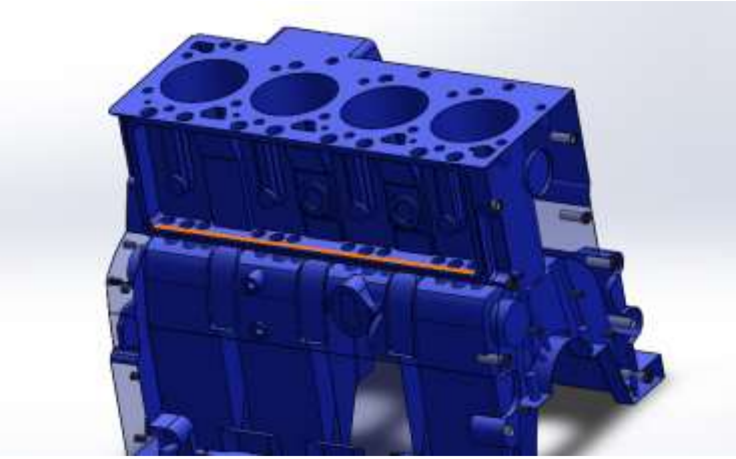


*Figure 3.3 Cylinder block of diesel engine*

The figure shown the cylinder block of marine diesel cummin engine 93 kw. In this case there is no jacket cooling system design. The jacket cooling system made by cylinder block dimension and manually measure in the laboratorium . The jacket cooling system design made using solidwork 3D.

Figure 3.4 is the dimention of  cylinder block. the lenghth is 515 mm and the width is 195 mm. From his dimention will be create the jacket cooling system in the cylincer block. the design of jacker cooling will create in solidwork 3D software.

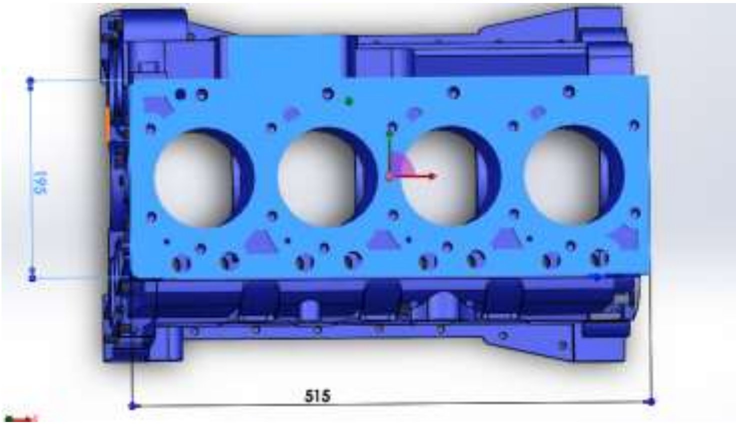



*Figure 3.4 Cylinder block dimention*

Figure 3.5 is the result of the jacket cooling system design with length 492mm and the height is 214mm. The design based on cylinder block dimention

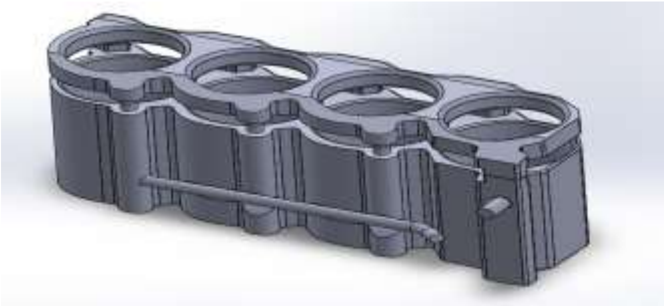


*Figure 3.5 jacket cooling design*

## 3.6 1D Cooling System Modeling

Cooling system modeling obtained from the existing data and needed to complete simulations and analysis in this thesis. Figure 3.5 is 1D modeling of cooling system in diesel engine

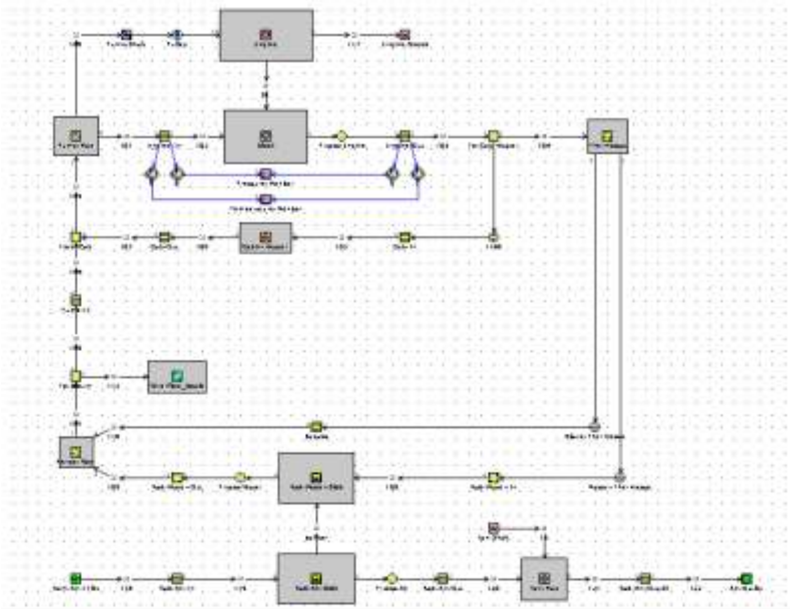


*Figure 3.6 1D Modeling of cooling system*

There is 8 cases in this simulation cooling system. The input of this simulation are RPM, fan speed, and temperature in the cylinder block. Table 3.1 is the case of the simulation cooling system based on Rpm and fan speed

*Table 3.1 engine speed and fan speed*

| Case | Rpm | Fan Speed |
|---|---|---|
| 1 | 1000 | 1000 |
| 2 | 1200 | 1200 |
| 3 | 1400 | 1500 |
| 4 | 1600 | 1700 |
| 5 | 1700 | 1900 |
| 6 | 1800 | 2000 |
| 7 | 2000 | 2200 |
| 8 | 2200 | 2200 |

Rpm of the engine based on engine performance with variation rpm. The minimum rpm is 1000 with minimum fan speed 1000 rpm and the maximum rpm of the engine is 2200 and the max fan speed is 2200 rpm. Fan speed data obtain from the figure 3.5

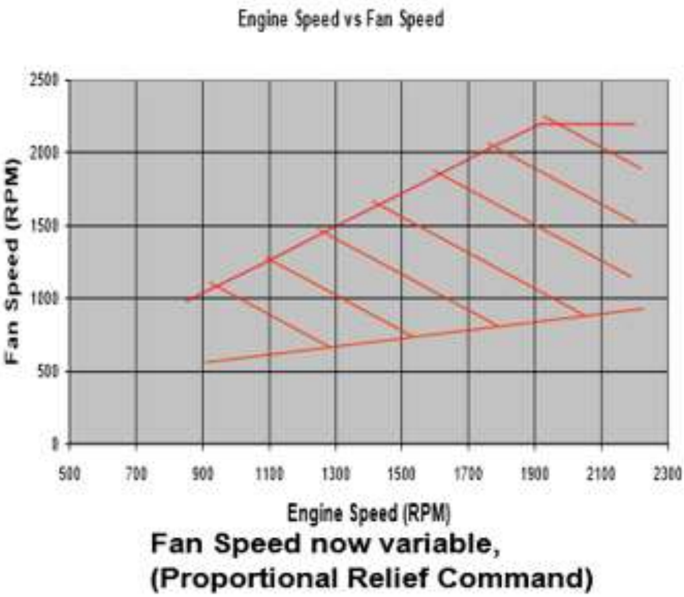



*Figure 3.7 engine speed vs fan speed graph*

The figure show the graph between fan speed (RPM) and the engine speed (RPM). If the engine speed is 1000, the range of the fan speed between 600 rpm until 1200 rpm. The fan speed 1000 rpm. The maximum engine speed is 2200 rpm and the fan speed range is higher than the lowest engine speed. The range of fan speed between 900 rpm until 2200 rpm. The fan speed will be assumed by 2200 rpm for the maximum rpm. the fan speed will affect the temperature of the jacket cooling in cylinder block. The engine speed increase will make the fan speed increase.

## 3.7 3D Jacket Cooling System Simulation

Before simulation of jacket cooling system, the design will be convert to parasolid for 3D simulation. There are some steps before doing the simulation.

### 3.7.1 Meshing of Jacket design

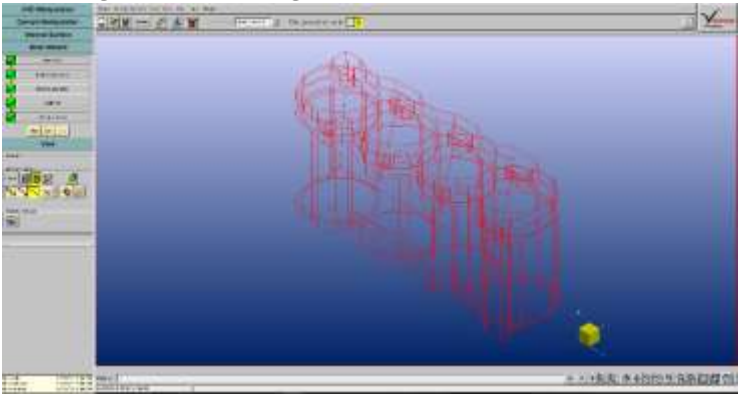


*Figure 3.8 meshing steps*

Figure 3.8 is the meshing process of jacket cooling system. A mesh is a network of line elements and interconnecting nodes used to model a structural system and numerically solve for jacket cooling system simulated behavior under applied loading. Computational technique create an analytical model by populating the material domain with a finite element mesh in which each line element is assigned mathematical attribute

Mesh size should be decreased through a series of analysis iterations until two subsequent sets of result are similar, or until balance between accuracy and analysis time are acceptable according to engineering criteria. Figure 3.9 is the initial mesh parameter used in software to meshing the jacket cooling design.

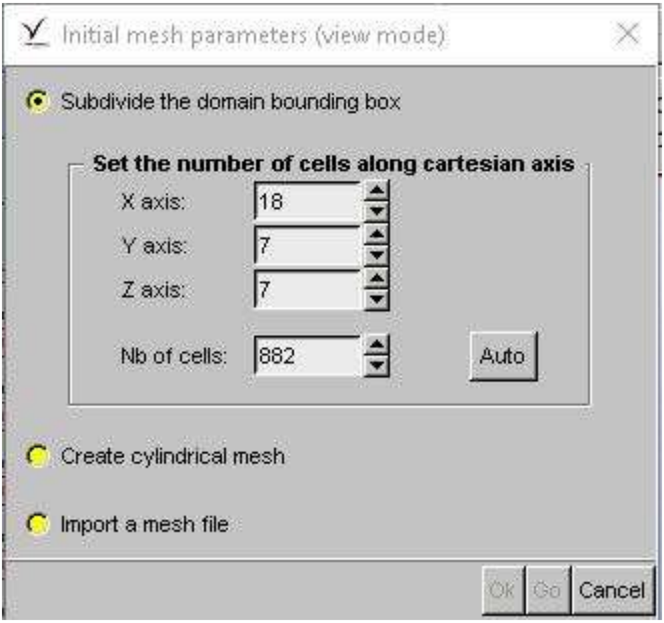



*Figure 3.9 input initial mesh parameters*

Figure 3.10 show the surface refinement. There are 3 tipes of surface of the jacket cooling system. There are Inlet, where the fluid flows into the jacket cooling system, outlet and wall of jacket cooling system

*Figure 3.10 surface refinement*

Table and table shows the optimization parameter input and Viscous layer insertion parameters to meshing the design of jacket cooling system

*Table 3.2 input optimization parameters*

| | |
|---|---|
| Max nb external optimization loops | 4 |
| Max nb of invalid cells | 100 |
| Max nb of final optimization iterations | 7 |
| Percentage of vertices to optimize during final optimization | 3.0 |
| Max nb of orthogonality optimization iterations | 0 |
| Minimal orthogonality threshold | 10 |

*Table 3.3 input optimization parameters 2*

| | |
|---|---|
| First layer thickness | 0.000208233 |
| Stretching ratio | 1.2 |
| Floating minimun number of layers | 5 |
| Floating maximum number of layers | 30 |

### 3.7.2 Physical Configuration

Table 3.4 is the input for jacket cooling 3D simulation. The domain configuration is fluid block and the time configuration is steady

*Table 3.4 input fluid model parameters*

| | | |
|---|---|---|
| Fluid name | Water | |
| Fluid type | incompressible | |
| Reference pressure | 101325.0 | Pa |
| Reference temperature | 330.0 | K |
| Cp | 4182.0 | J/kg K |
| Heat conduction law | Prandtl | |
| Prandtl number | 7.02 | |
| Viscosity law | Constant viscosity | |
| Kinematic viscosity | 1.01e-006 | m2/s |
| Density law | Boussinesq | |
| Density | 1001.0 | Kg/m3 |
| compressibility | 1e-011 | 1/Pa |
| dilatation | 0.000206 | |
| Reference length | 1.03 | m |
| Reference velocity | 2.0 | m/s |
| Reynold number | 2.0396E+006 | |
| Flow model | Turbulent navier stokes | |

Figure 3.11 is the heat source in jacket cooling system. Heat source came from the cylinder liner. There are 4 cylinders in cummin diesel engine which makes 4 heat source with the same value. The heat flux is constant with value 52410.0 W

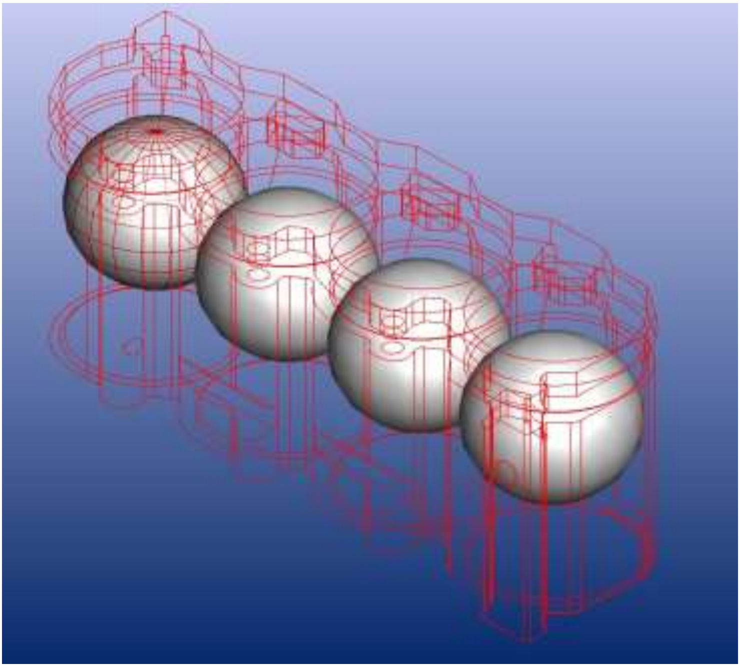


*Figure 3.11 heat source*

### 3.7.3 Boundary Condition

Table is the boundary condition. Boundary condition define the inputs of the simulation model. Some condition, like velocity, pressure, and temperature, define how a fluid enters and leave the jacket cooling system design. Boundary condition connect the simulation model surroundings. Mass flow input and output same because using continuity law

*Table 3.5 input boundary condition parameters*

| | | |
|---|---|---|
| Inlet | | |
| Mass flow | 2.45 | Kg/s |
| Static temperature | 330 | K |
| K | 9.375 | m2/s2 |
| epsilon | 9219.6 | m2/s3 |
| Outlet | | |
| Mass flow | 2.45 | Kg/s |
| Initial pressure | 101325 | Pa |
| Solid | | |
| Static temperature | 567 | K |

## 3.8 Validation

At this stage, the output data from simulation process not exactly resulted the correct data of engine performance and the cooling system analysis. The output data obtained as a result from engine simulation might be had some of error. So we have to repeat the step of the simulation stage to find failures and corrected the input data. If the data has been obtained correctly, then we can go to the next step.

## 3.9 Conclusion and Reporting

The conclusion based on the result of the analysis. What can we get from the analysis, put in the conclusion and reporting, Conclusions are expected in this thesis is able to answer the problem. Written advice based on data from the discussion well as the fact that there is, and given to the improvement of this bachelor thesis in order to become better.

# CHAPTER 4

# DISCUSSION AND RESULTS

In this following chapter will explain the simulation results of 1D cooling system modeling as a result is mass flow rate and average of temperature of fluid a cylinder block. The result of the modeling will use to simulate the 3D jacket cooling system analysis as the result is flow and heat transfer in jacket cooling system

## 4.1 Information Data Used

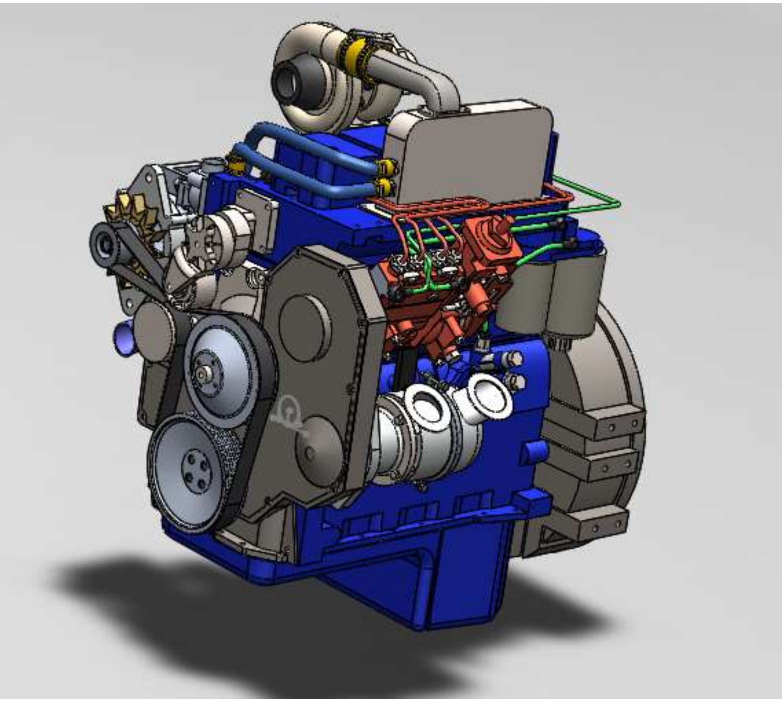


*Figure 4.1 Diesel Engine 93KW*

1. Brand : Cummins
2. Engine Model : 4BTA3.9-M
3. No. of Cylinder : 4
4. Rated Power : 125 BHP (93 kW) @2200 rpm
5. Displacement : 3.9 L
6. Bore : 102 mm
7. Stroke : 120 mm
8. Inlet Valve : 45 mm (clearance = 0.25 mm)
9. Outlet Valve : 43 mm (clearance = 0.51 mm)
10. Compression Ratio : 16.5 : 1
11. Firing Order : 1-3-4-2
12. Exhaust System

- Maximum Back Pressure : 76 mmHg
- Exh. Pipe Normally Acceptable : 75 mm

13. Air Intake System

- Max. Intake Air Restriction with Heavy Duty Air Cleaner

- Clean Element : 381 mmH2O
- Dirty Element : 635 mmH2O

- Minimum Dirt Holding Capacity with Heavy Duty Air Cleaner : 53g/litre/sec
- MaximumTemperature Rise from Ambient to the Inlet of the Turbocharger : 17 ℃

14. Lubrication System

- Normal Operating Oil Pressure Range : 69~345 kPa
- Maximu Lube Oil Flow for Engine : 4 litre/min
- Maximum Sump Oil Temperature : 127 ℃
- Minimum Engine Oil Pressure for Engine Protection Devices

- At Rated Speed and Load : 276 kPa
- At Torque Peak Speed and Load : 207 kPa
- At Low Idle : 69 kPa

- Minimum Required Lube System Capacity - Sump plus Filters : 9 litres

15. Fuel System

- Maximum Fuel Flow on the Supply Side of the Fuel Pump

- With clean fuel filter : 102 mmHg

- With dirty fuel filter : 203 mmHg

- Maximum Fuel Drain Restriction

- With check valves : N/A

- Less check valves : 510 mmHg

- Maximum Fuel Inlet Temperature : 71 ℃S

16. Low Idle Set Speed : 750 rpm
17. Maximum Governed Speed : 2600 rpm

Maximum Over speed Capability : 3750 rpm

## 4.2 Cooling System Modeling

Figure 4.2 is the result of the modeling cooling system in diesel engine

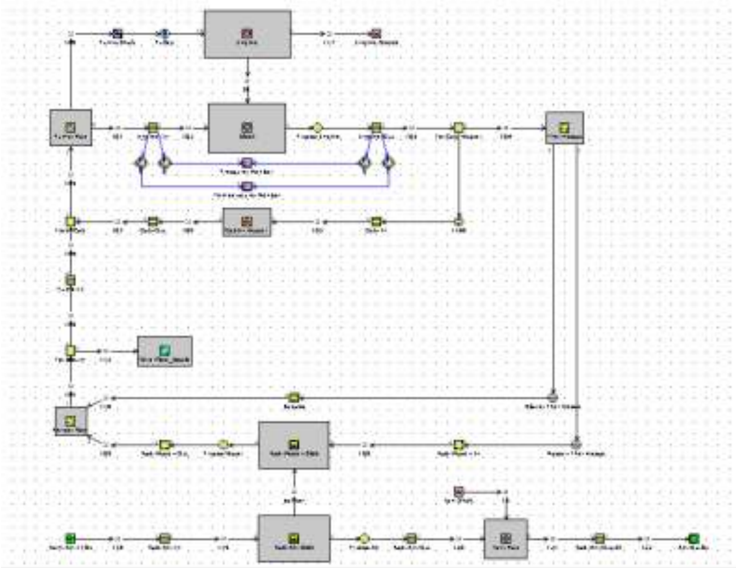


*Figure 4.2 cooling system modeling*

From the analysis of the modeling the output will be in cylinder block such as average temperature, mass flow rate, heat transfer. The analysis would be in 8 cases from the lower engine speed until higher rpm. The range are between 1000 – 2200 rpm.

Table 4.1 is a simulation result of the 1D cooling system modeling. In according with the result of the simulation the static data been obtain a result of temperature and mass flow rate of the cooling system in Engine block.

*Table 4.1 output data cylinder block*

| NO | Part Name | Block |
|---|---|---|
| 1 | Mass flow rate (g/s) | 2452,4 |
| 2 | Fluid temperature (K) | 536,9 |
| 3 | Wall Temperature (K) | 567,1 |
| 4 | Heat Transfer (kW) | 52,41 |

From table 4.1 showing the result of the cooling system in cylinder block with 2200 rpm. All the result are output of cooling system in cylinder block. For the output mass flow rate is 2452,4 g/s, fluid temperature after through the Engine block is 536,9 K and the wall temperature or cylinder block temperature is higher than fluid temperature is 567,1. And the heat transfer happen in between cooling fluid and the cylinder block wall is 52,41 KW. All this result will use to simulate the flow in 3D modeling of jacket cooling system.

The figure 4.3 and table 4.2 is the radiator water side and radiator air side modeling of cooling system and the output of the radiator

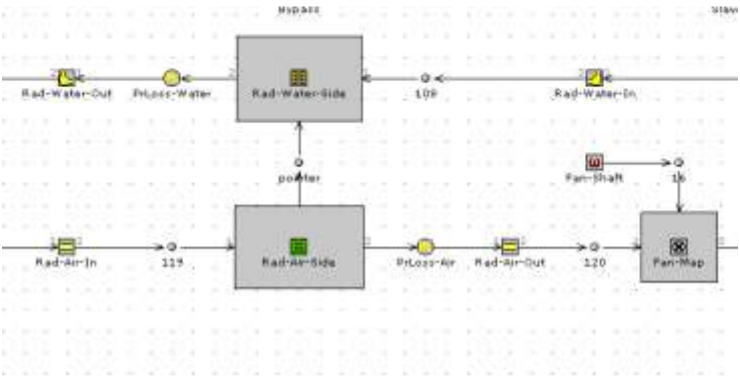



*Figure 4.3 water and air radiator modeling*

*Table 4.2 output data radiator*

| Part Name | Rad-water-side | Rad-air-side |
|---|---|---|
| Mass flow rate (g/s) | 2015.6 | 244.2 |
| Volumentric flow rate (litre/s) | 2.0 | 279.1 |
| Fluid temperature (K) | 530.1 | 498.3 |
| Wall temperature (K) | 504.4 | 504.4 |
| Pressure (bar) | 1.1 | 1.0 |
| Density (kg/m$^3$) | 1000.68 | 0.70 |
| Heat transfer (kW) | -49.42 | 49.42 |
| Effectiveness (%) | 82.27 | 82.27 |
| Volume (litre) | 2.00 | 16.00 |
| Transfer units | 1.82 | 1.82 |
| Flow Capacity ratio | 0.03 | |

## 4.3 Fluid flow Jacket Cooling system 1400 Rpm

Table 4.3 is the input for jacket cooling system 3D simulation with engine speed 1400 rpm

*Table 4.3 Input for 3D simulation 1400 rpm*

| NO | Part Name | Block |
|---|---|---|
| 1 | Mass flow rate (g/s) | 2452,4 |
| 2 | Fluid temperature (K) | 433.8 |
| 3 | Wall Temperature (K) | 468.2 |
| 4 | Heat Transfer (kW) | 59,68 |

The figure 4.4 is the result of jacket 3D simulation as the output is velocity. The figure show that the velocity is high when the fluid enters the jacket cooling system. the velocity is high because there is force by pump which make the velocity high when in the inlet area. When the fliud already in the jacket cooling, the velocity will decrease. There are some space in various area in jacket cooling system, in that area there are a fluid with the velocity is very low and there is turbulent flow. Flow in which the fluid undergoes irregular fluctuations, or mixing, in which the fluid moves in smooth paths or layer. Turbulent flow make is the best flow for heat transfer process and higher efficiency of heat transfer.

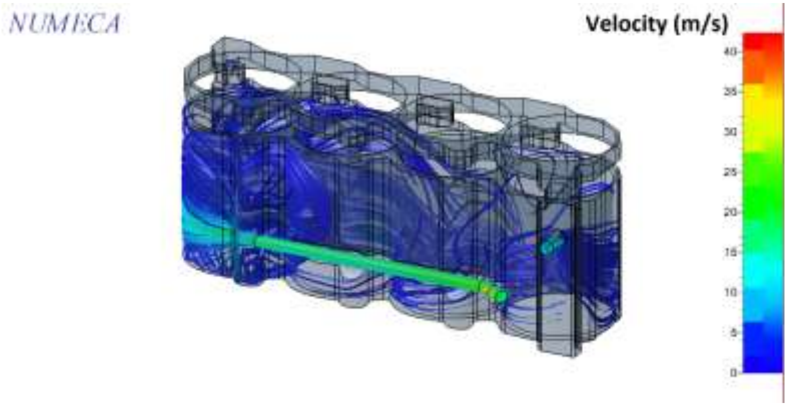



*Figure 4.4 velocity Fluid flow in jacket cooling (1400 rpm)*

The velocity in outlet is higher than velocity in the jacket cooling and it will same velocity as inlet because of continuity equation. In fluid dynamic, the continuity equation state that, in any steady state process, the rate at which mass enters a jacket cooling system is equal to the rate at which mass leaves the jacket cooling system. the figure 4.5 show the other side of fluid flow in jacket cooling system

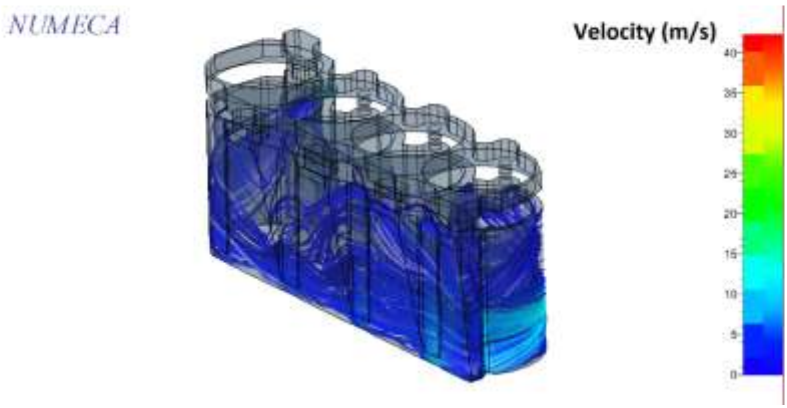



*Figure 4.5 velocity fluid flow in jacket cooling (2)*

Figure 4.6 is the result of temperature flow in jacket cooling system. before fluid enters the jacket cooling, the temperature of the fluid is low around 300 – 350 K. When the fluid enters tha jacket cooling, it will be increase the temperature of the fluid flow. The temperature of the fluid increase because of the wall temperature. Heat treansfer will happen in jacket cooling system between wall temperature and fluid temperature. The fluid make absorb the high temperature in wall of jacket cooling and make the fluid increase the temperature, it means the outlet temperature of fluid will be higher than before the fliud enters the jacket cooling system.

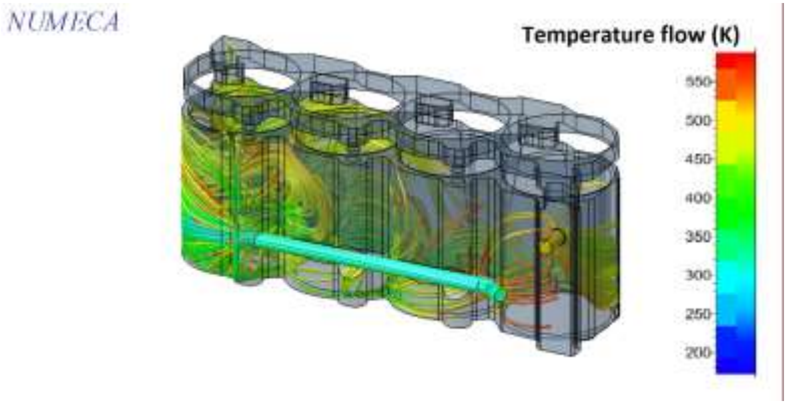



*Figure 4.6 temperature flow in jacket cooling*

## 4.4 Fluid flow Jacket Cooling system 1800 Rpm

Table 4.4 is the input for jacket cooling system 3D simulation with engine speed 1800 rpm

Table 4.4 *input for 3D simulation 1800 rpm*

| NO | Part Name | Block |
|---|---|---|
| 1 | Mass flow rate (g/s) | 2452,4 |
| 2 | Fluid temperature (K) | 468.5 |
| 3 | Wall Temperature (K) | 501.6 |
| 4 | Heat Transfer (kW) | 59,68 |

Figure 4.7 is the result of the fluid flow jacket cooling system with 1800 rpm. The figure show that the velocity is high when the fluid enters the jacket cooling system. the velocity is high because there is force by pump which make the velocity high when in the inlet area. The vector line is more than the lowest engine speed

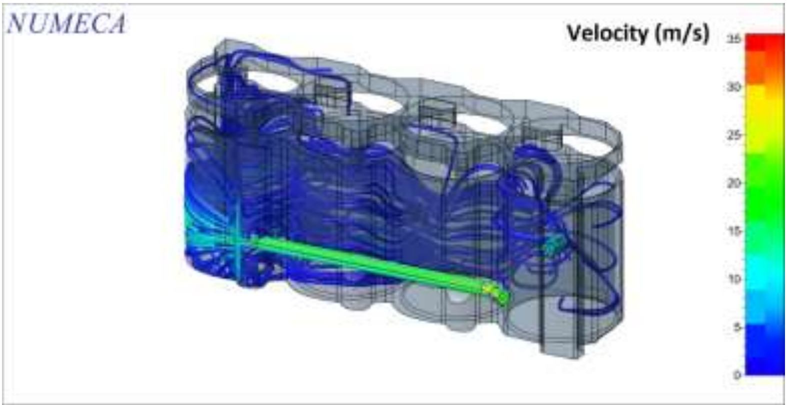



*Figure 4.7 flow velocity with 1800 rpm*

Figure 4.8 is the result of the flow temperature in jacket cooling system. the temperature is increasing when trought the jacket cooling because of heat transfer in jacket cooling system

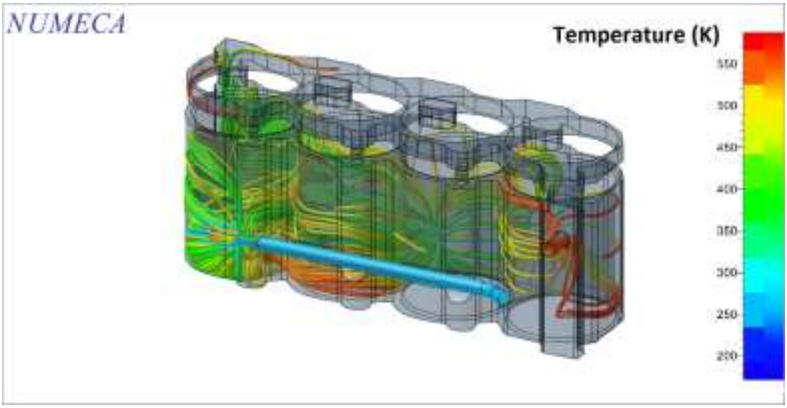



*Figure 4.8 temperature flow with 1800 rpm*

## 4.5 Fluid flow Jacket Cooling system 2200 Rpm

Table 4.5 is the input for jacket cooling system 3D simulation with engine speed 2200 rpm

*Table 4.5 input for 3D simulation 2200 rpm*

| NO | Part Name | Block |
|---|---|---|
| 1 | Mass flow rate (g/s) | 2452,4 |
| 2 | Fluid temperature (K) | 536,9 |
| 3 | Wall Temperature (K) | 567,1 |
| 4 | Heat Transfer (kW) | 52,41 |

Figure 4.9 is the flow velocity of jacket cooling system when engine speed is 2200 rpm. The vector line of the simulation is more that the lowest engine speed. In the first cylinder (from left), the vector line almost fill all in first cylinder because the velocity of fluid flow is high

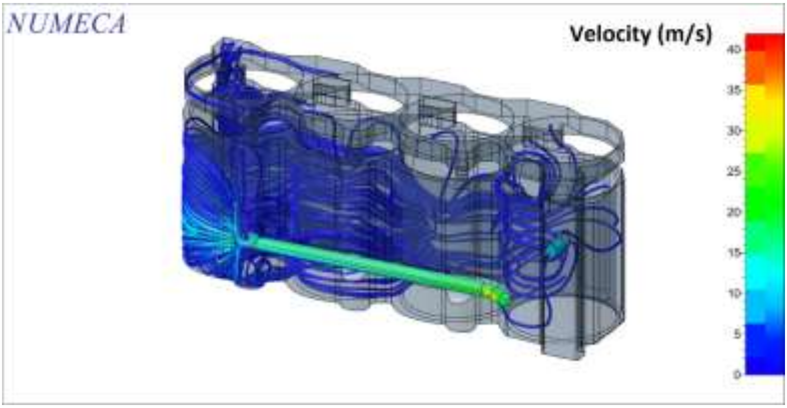



*Figure 4.9 flow velocity with 2200 rpm*

Figure 4.10 is the temperature flow of jacket cooling system when the engine speed reach the maximum rpm. The engine speed for this simulation is 2200 rpm. The temperature is increasing after throught the jacke cooling system.

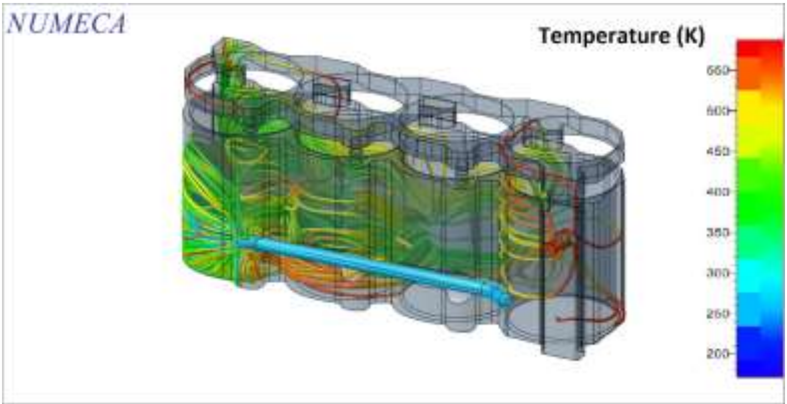



*Figure 4.10 temperatrure flow with 2200 rpm*

## 4.6 Efficiency of jacket cooling system

Efficiency of jacket cooling system obtain when the 3D simulation was done. The efficiency result of jacket cooling system will be in graph.Figure 4.11 is the efficiency of jacket cooling system with 1400 rpm. The cycle is between 0-1000.

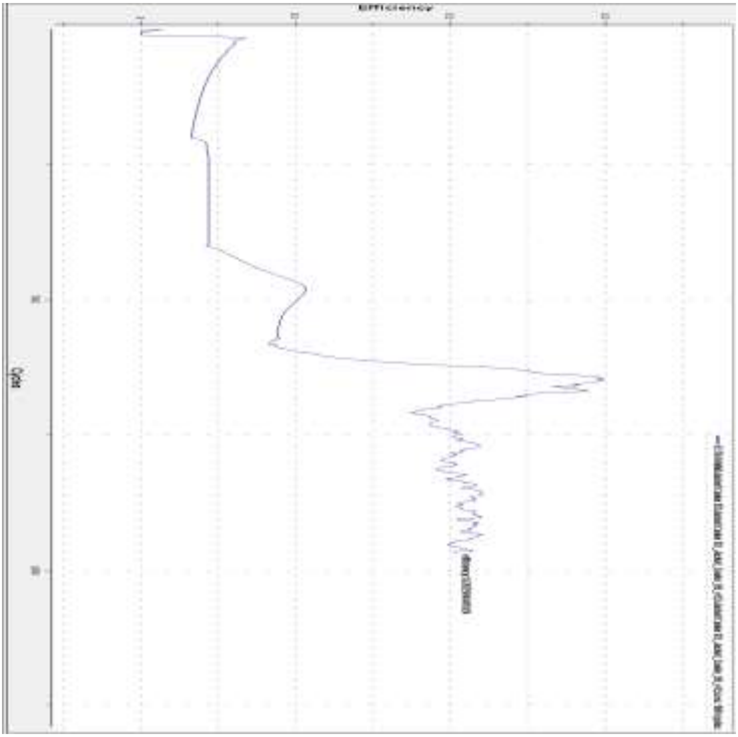


*Figure 4.11 efficiency of jacket cooling with 1400*

Figure 4.12 is the efficiency of jacket cooling when the engine speed is 1800 rpm. The cycle is between 0-1000.

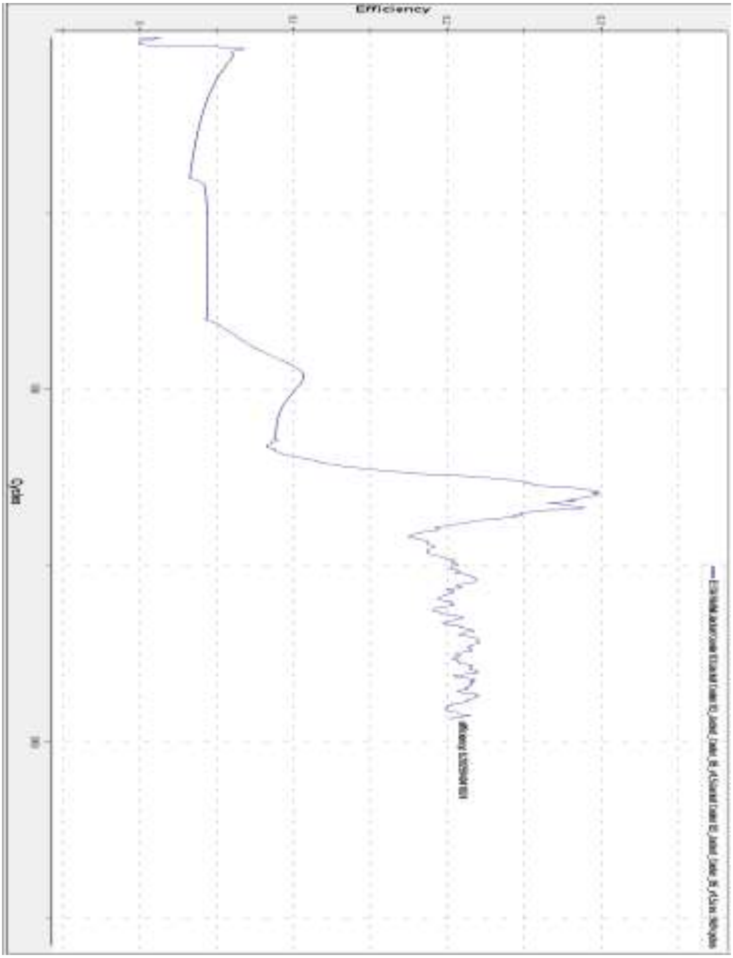



*Figure 4.12 efficiency of jacket cooling with 1800 rpm*

Figure 4.13 is the efficiency of jacket cooling when the engine speed is 2200 rpm. The cycle is between 0-500

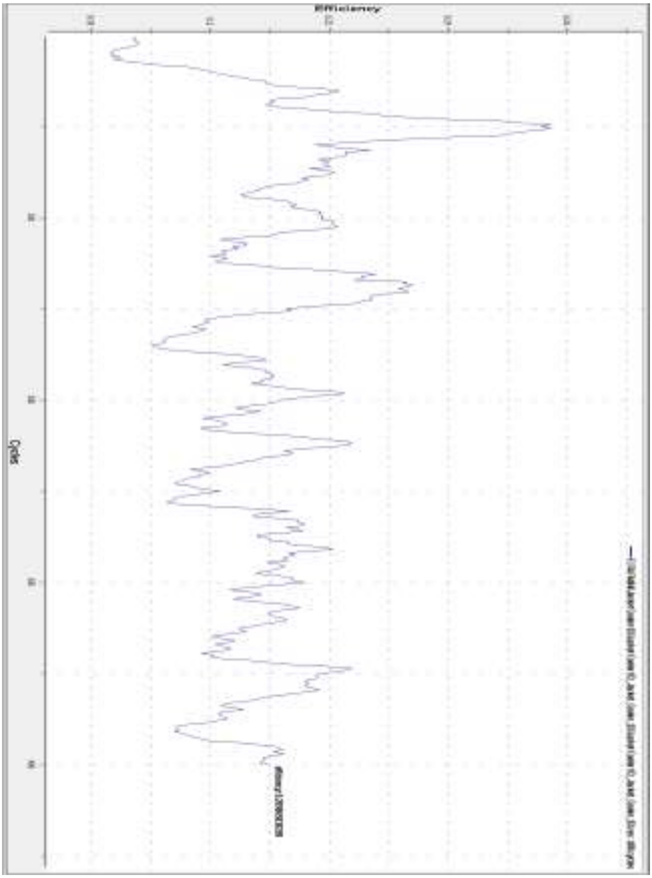



*Figure 4.13 efficiency of jacket cooling with 2200 rpm*

*"This Page Intentionally Left Blank"*

# CHAPTER 5
# CONCLUSION AND SUGGESTION

## 5.1 Conclusion

From the result of research analysis it was concluded that:

1. The simulation begin with the design of jacket cooling system. The design made based on cylinder block and cylinder head dimention. The design have an intlet and oulet to show the fluid when enters the jacket cooling until leaves the jacket. the data will be used in fluid flow jacket cooling system simulation are mass flow, temperature of fluid and wall temperature of jacket cooling and heat transfer. All the data will obtained by simulate the 1D modeling with 1D simulation software. The data will be required to show the fluid flow and the temperature flow in the jacket cooling system using numeca software
2. With the jacket cooling design in marine diesel engine 93 KW, the flow in jacket cooling system is good. But in this design there are much turbulent flow in jacket cooling system and it can make the efficiency of jacket cooling system is low. from the simulation, the higest engine speed of engine will increasing the velocity of fliud which enters the jacket cooling system beacause the cylinder block of engine need more fluid flow for cool down the cylinder block and the lowest engine speed will make the velocity is decreasing because the cylinder block temperature is lowest than cylinder block with higest engine speed

## 5.2 Suggestion

The jacket cooling system design can use in marine diesel engine 93 KW with the same dimention,but the fluid flow in jacket cooling system will be better if the inlet and outlet be change the position. The inlet will be lower and the oultet will be higher, it will can make a diffrent fluid flow in jacket cooling system. it can be better if there are inlets and outlets at each cylinder, it means there are 4 inlets and 4 outlets. The flow will reach all the area of jacket cooling system

# REFERENCES

Gamma Technologies, 2009. GT-POWER User's Manual Version 7.0, Gamma Technologies Inc.

A.M. Kriger, M.E. Diskin, M.E. Diskin, Liquid Cooling of Automotive Engine, Mashinostroenie, Moscow, 1985.

A.A. Zhilin, A.B. Zharov, Heat Transfer Numerical Modelling for IC Engine Coolant Jacket, in: Proceeding of International scientific symposium Avtotraktorostroenie – 2009, Moscow, MAMI. (2009) 78-88. URL: http://www.mami.ru/science/autotr2009/scientific/article/s02/s02_13.pdf

V.A Romanov Experience of the Diesel Engine Cooling System Simulation Volume 150, 2016, Pages 490–496

Mulemane, A., Soman, R.,2007, "CFD Based Complete Engine Cooling Jacket Development and Analysis", SAE 2007-01-4129.

J. H. Ferziger and M. Peric, Computational Methods for Fluid Dynamics. Springer, 1996.

P. Wesseling, Principles of Computational Fluid Dynamics. Springer, 2001.

Mohammadi A and Yag houbi, 2011, Two Phase Flow Simulation for Subcooled Nucleat Boiling Heat Transfer Calculation in Water Jacket of Diesel Engine, Vol 2, Spring 2011

Raharjo Juniono, 2011, Design marine diesel using Reverse Engineering Method

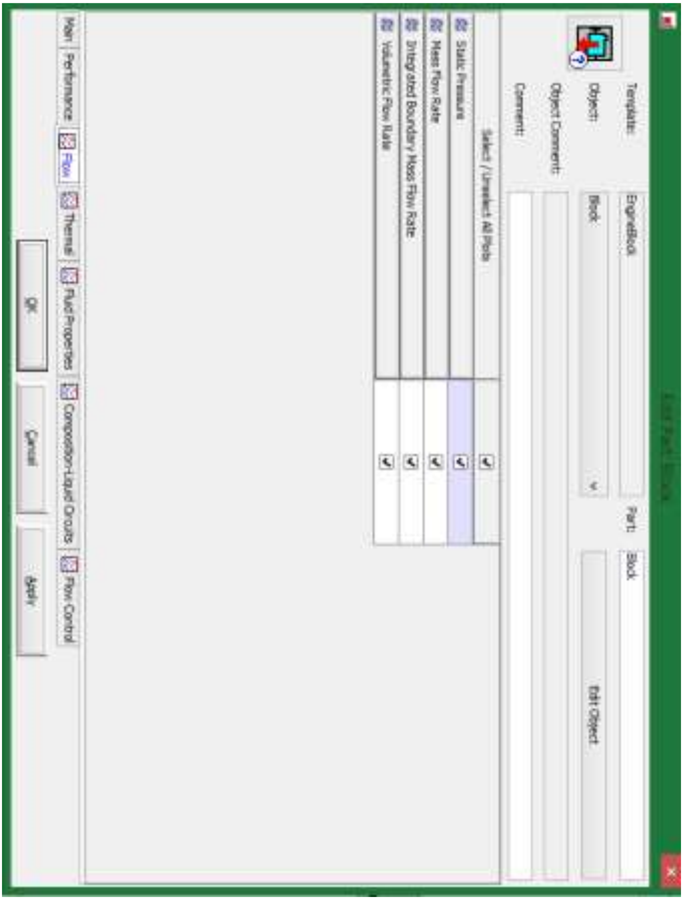
Alexey Vdovin, 2010, Cooling performance simulations in GT-Suit, Chalmers Reproservice, Göteborg, Sweden 2010

U.S Department of Energy, Thermodynamic, Heat Transfer, and Fluid Flow, Volume 1-3, Washington, June 1992

# APPENDIX

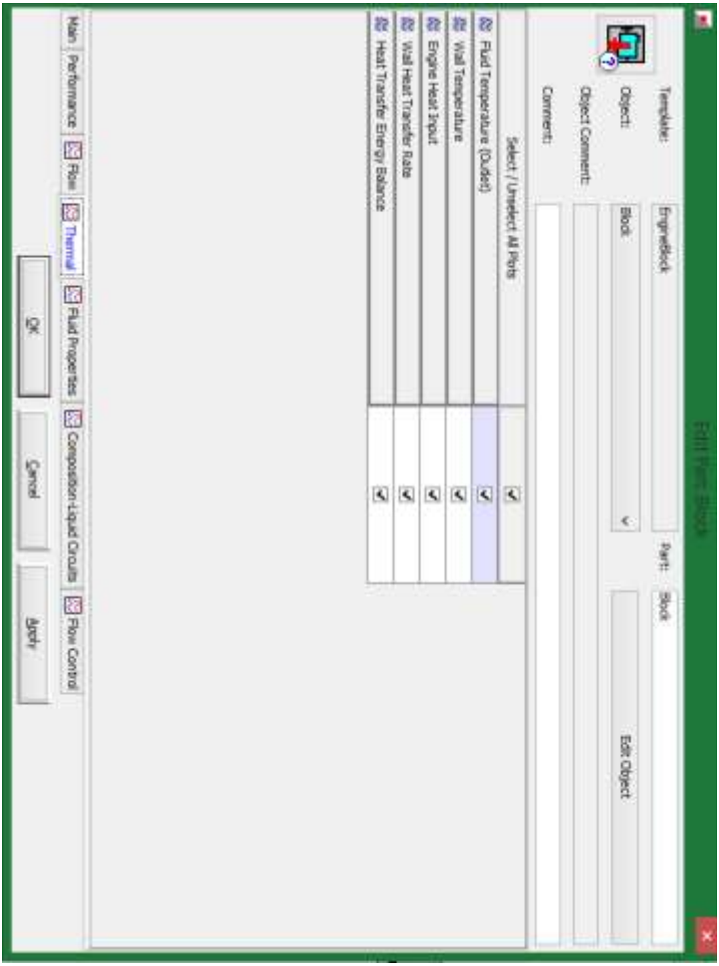
## ATTACHMENT 1

Output flow of the cooling system will show



**ATTACHMENT 2**

Output thermal of the cooling system will shown



**ATTACHMENT 3**

Nusselt Number Correlation : Rad_Experimental of heat exchanger with 2200 rpm

| | Stream #1 | Stream #2 |
|---|---|---|
| Laminar Re Number Limit | 7628 | 10648 |
| Transition Re Number Limit | 7628 | 10726 |
| Laminar Exponent | 0.8110 | 0.9241 |
| Transition Exponent | 0.7419 | 0.9803 |
| Turbulent Exponent | 0.8473 | 0.2909 |
| Turbulent Coefficient | 0.0328 | 146.3830 |

**ATTACHMENT 4**

Heat exchanger engine block output data with 2200 rpm

| Part Name | Block |
|---|---|
| Mass Flow Rate [g/s] | 2452.4 |
| Fluid Temperature [K] | 536.9 |
| Wall Temperature [K] | 567.1 |
| Heat Transfer [kW] | 52.41 |

## ATTACHMENT 5

Heat exchanger pair output data in radiator water side and radiator air side with 2200 rpm

| | MASTER | SLAVE |
|---|---|---|
| Part Name | Rad-Water-Side | Rad-Air-Side |
| Heat Transfer Object | Rad_Experimental | Rad_Experimental |
| Mass Flow Rate [g/s] | 2015.6 | 244.2 |
| Volumetric Flow Rate [liter/s] | 2.0 | 279.1 |
| Fluid Temperature [K] | 530.1 | 498.3 |
| Wall Temperature [K] | 504.4 | 504.4 |
| Pressure [bar] | 1.1 | 1.0 |
| Density [kg/m^3] | 1000.68 | 0.70 |
| Heat Transfer [kW] | -49.42 | 49.42 |
| Effectiveness [%] | 82.27 | 82.27 |
| Volume [liter] | 2.00 | 16.00 |
| Transfer Units | 1.82 | 1.82 |
| Flow Capacity Ratio | 0.03 | 0.03 |

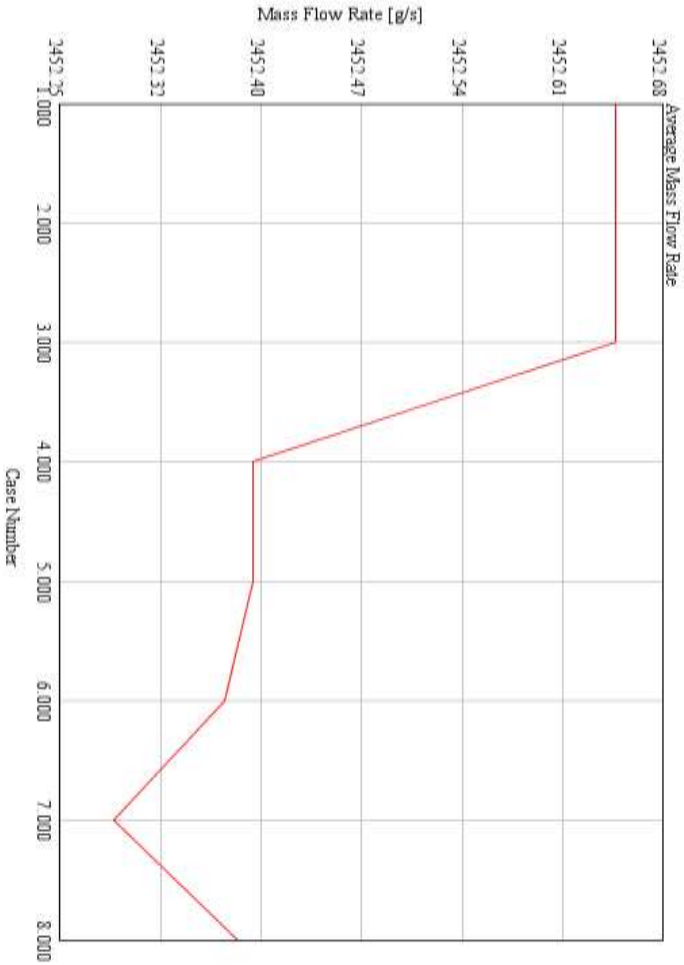
# ATTACHMENT 6

HxNuCorr Objects output data 2200 rpm

| Object Name | Rad_Experimental_int | Rad_Experimental_ext |
|---|---|---|
| Laminar Re Number Limit | 7628 | 10648 |
| Transition Re Number Limit | 7628 | 10726 |
| Laminar Exponent | 0.8110 | 0.9241 |
| Transition Exponent | 0.7419 | 0.9803 |
| Turbulent Exponent | 0.8473 | 0.2909 |
| Turbulent Coefficient | 0.0328 | 146.3830 |
| Ref. Length [m] | 0.01 | 0.01 |
| Heat Transfer Area [m^2] | 0.50 | 1.50 |
| Flow Area [m^2] | 0.01 | 0.20 |

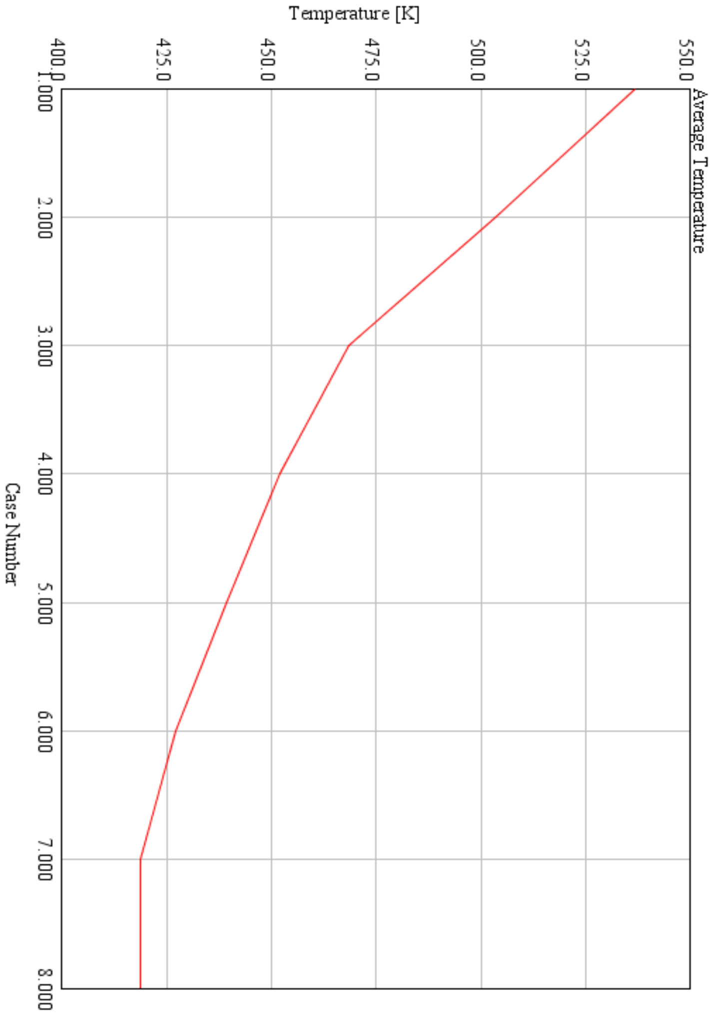
**ATTACHMENT 7**

Graph Mass flow rate of water cooling system based on case number (engine speed 1000-2200 rpm)



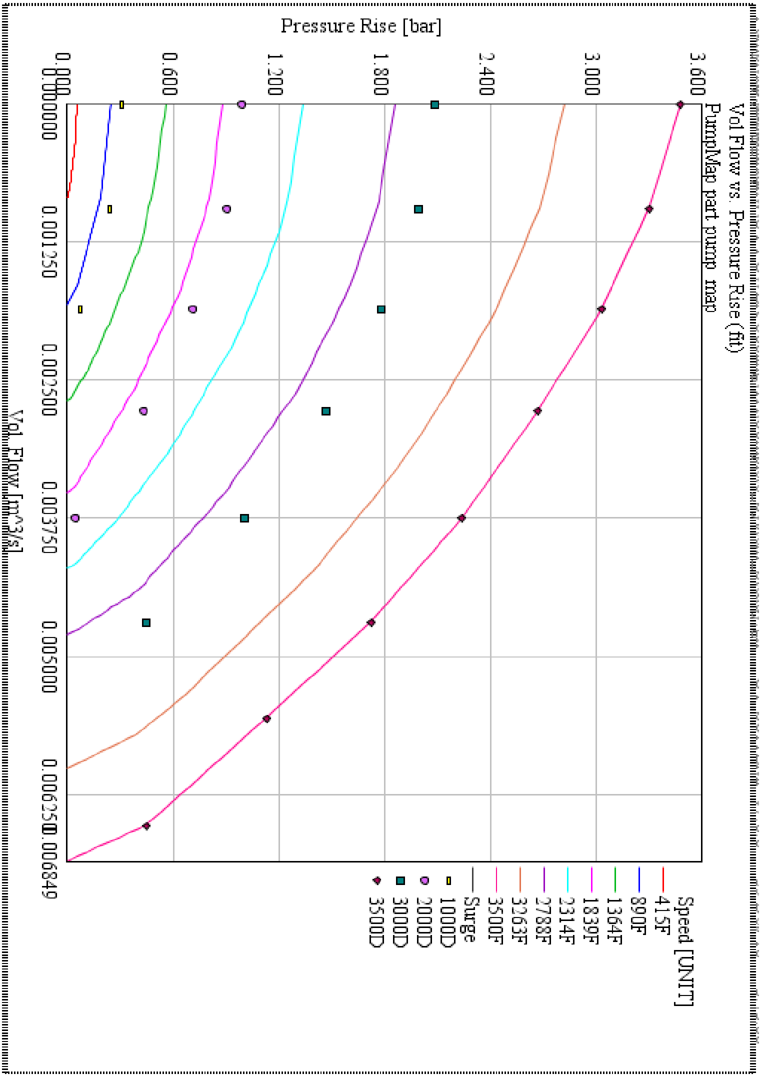
**ATTACHMENT 8**

Graph Average temperature based on case number (engine speed)



# ATTACHMENT 9

Pump Map



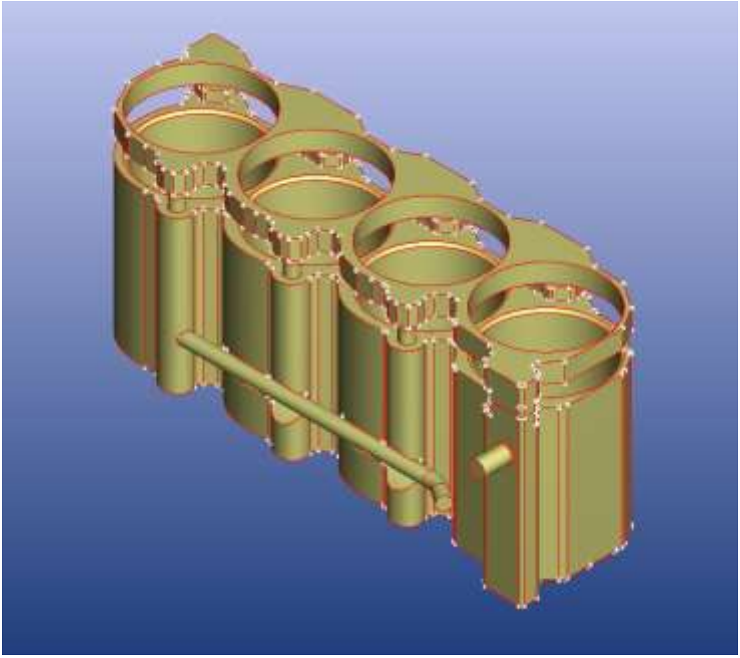
## ATTACHMENT 10

Mechanical Model information

| | |
|---|---|
| Number of Mechanical Components | 7 |
| Number of Mechanical Connections | 5 |
| Number of Elements | 9 |
| Number of Nodes | 5 |
| --- Number of Structural Nodes | 3 |
| ------ Reference Nodes | 3 |
| ------ Connecting Nodes | 0 |
| --- Number of Slave Nodes (1) | 1 |
| --- Number of Internal Nodes (2) | 2 |
| Solution Matrix Size | 7 |
| --- Equations of Motion (dofs) | 1 |
| --- Constraint Equations | 6 |
| Number of Matrix (skyline) Elements | 43 |
| Average Matrix (skyline) Bandwidth | 2.57 |

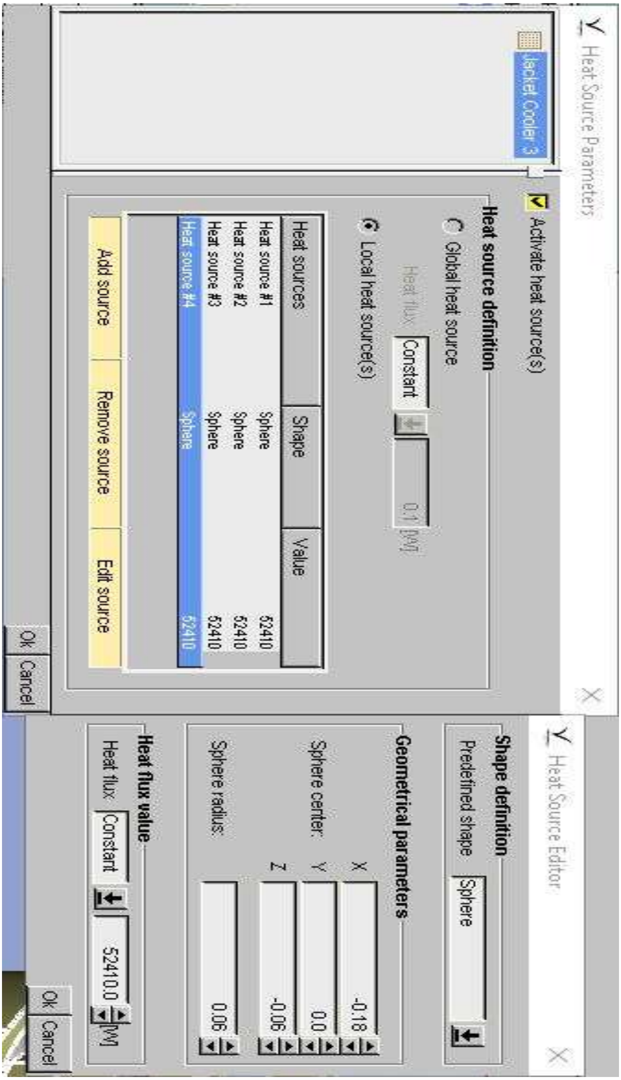
**ATTACHMENT 11**

Fillet on jacket cooling design



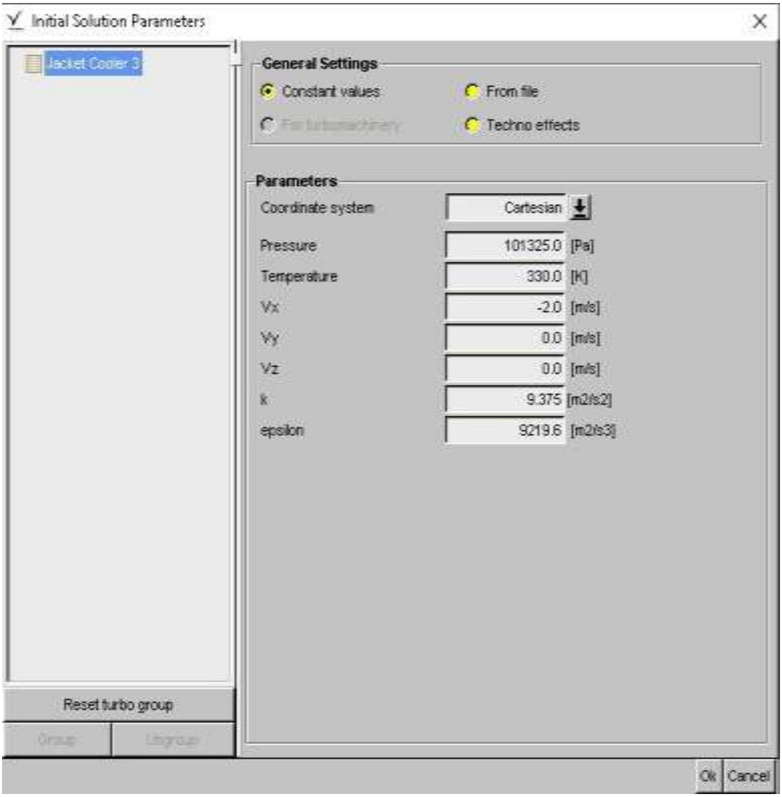
# ATTACHMENT 12

Input Heat source parameter



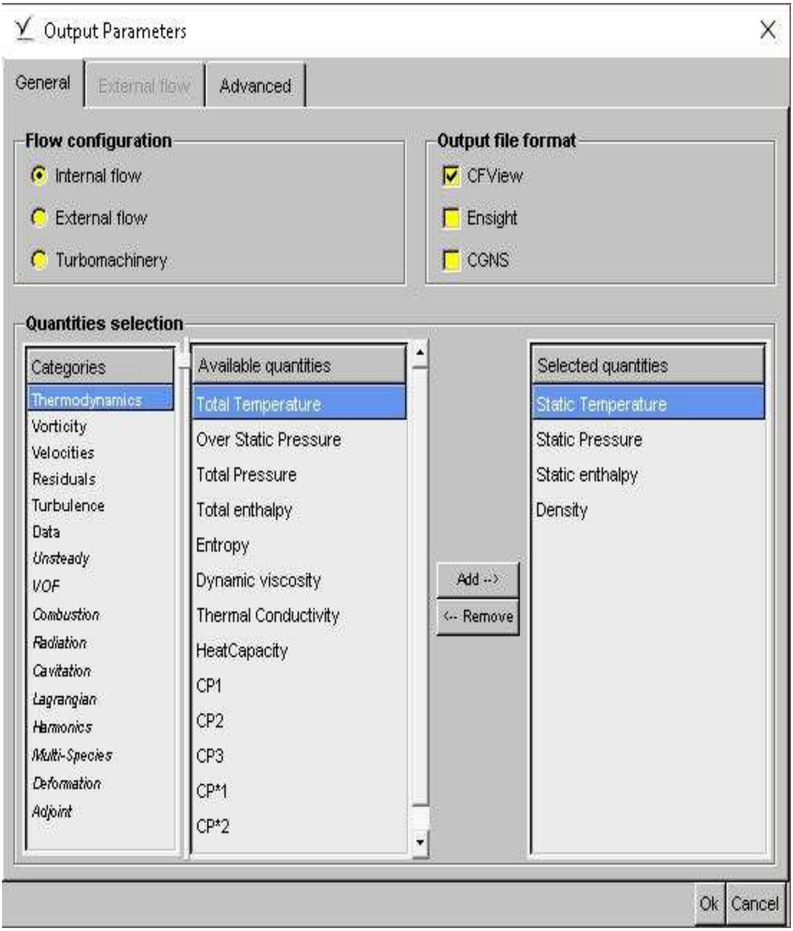
# ATTACHMENT 13

Input Initial solution parameter of jacker cooler



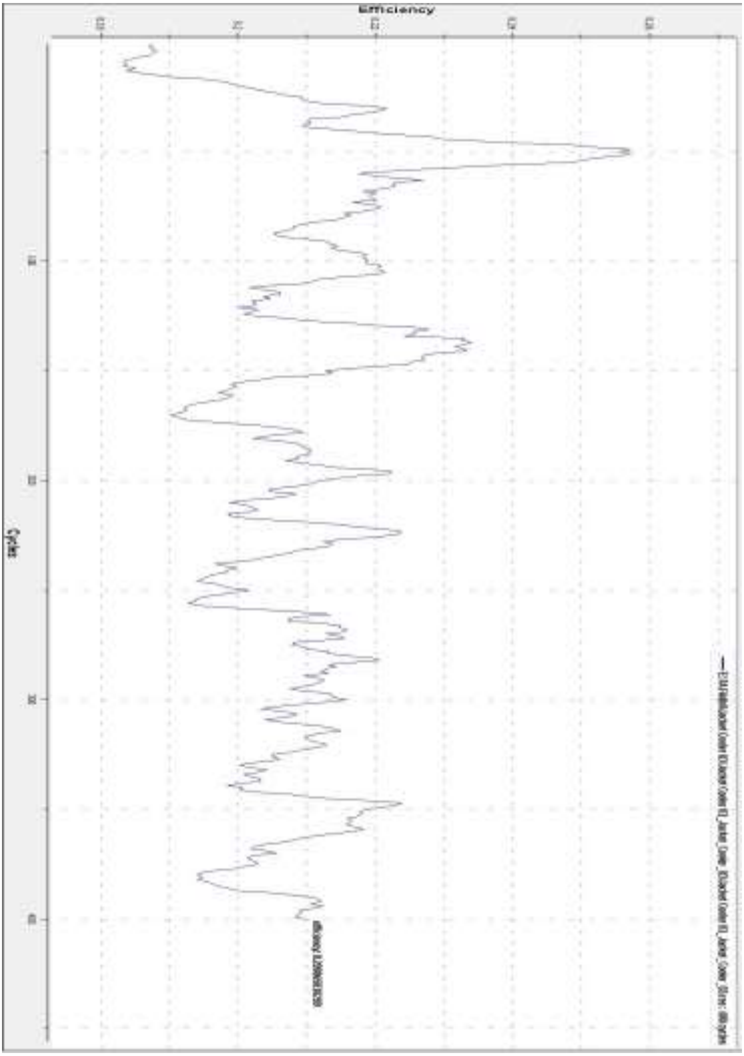
# ATTACHMENT 14

Output parameters of jacket cooling system



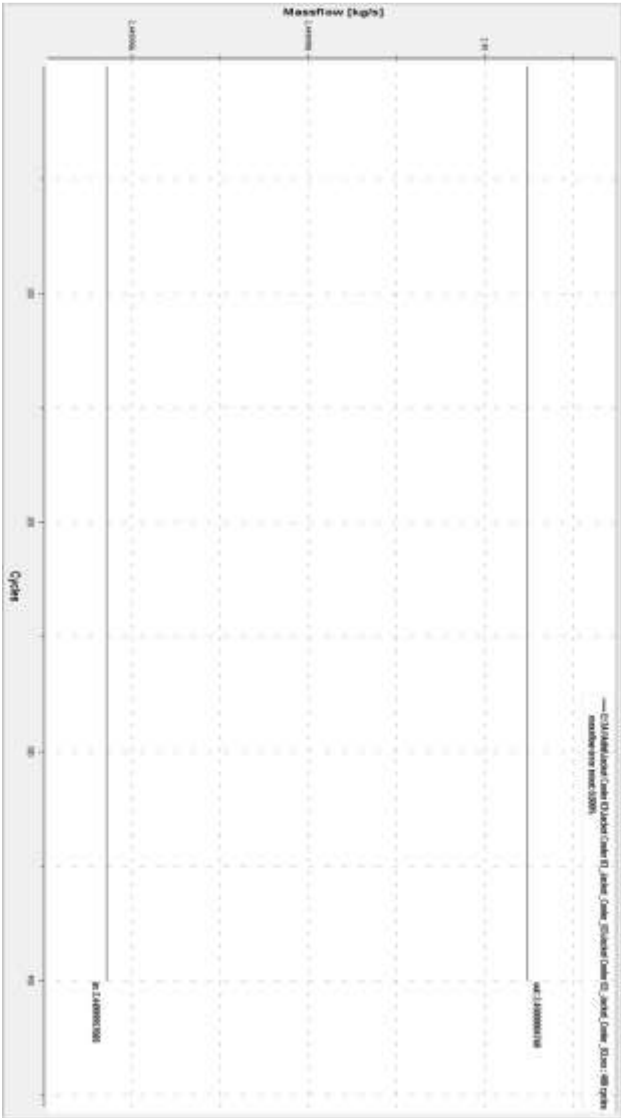
**ATTACHMENT 15**

Efficiency of jacket cooling system



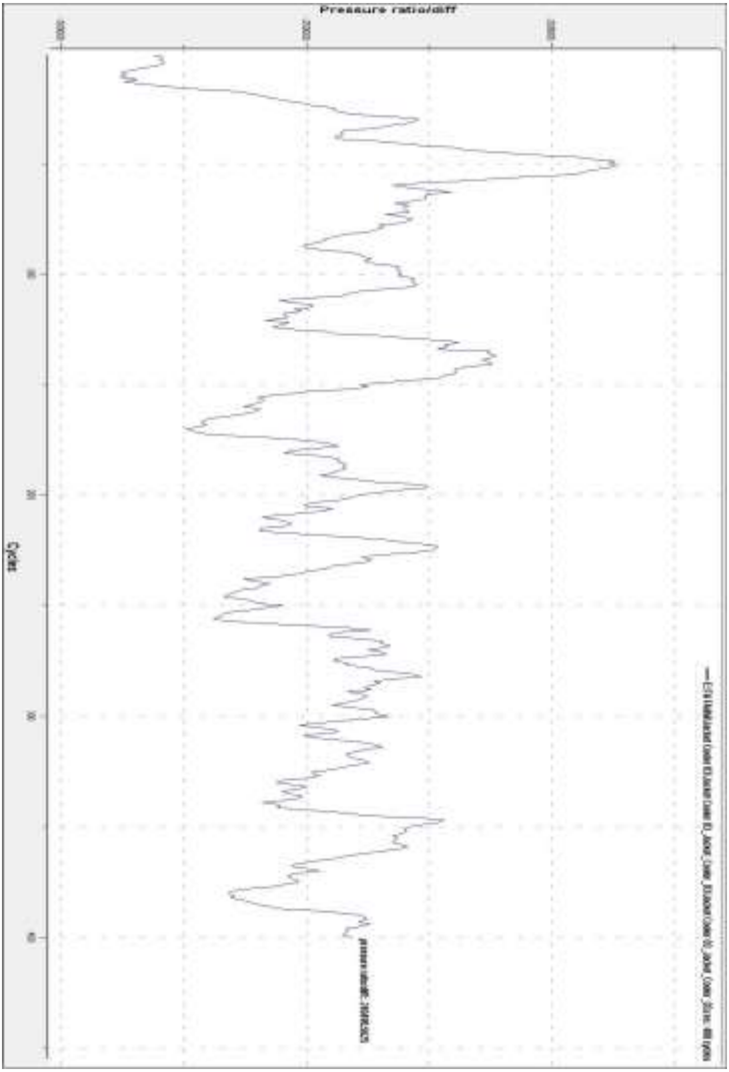
# ATTACHMENT 16

Mass flow rate of fluid flow in jacket cooling system (2200 rpm)



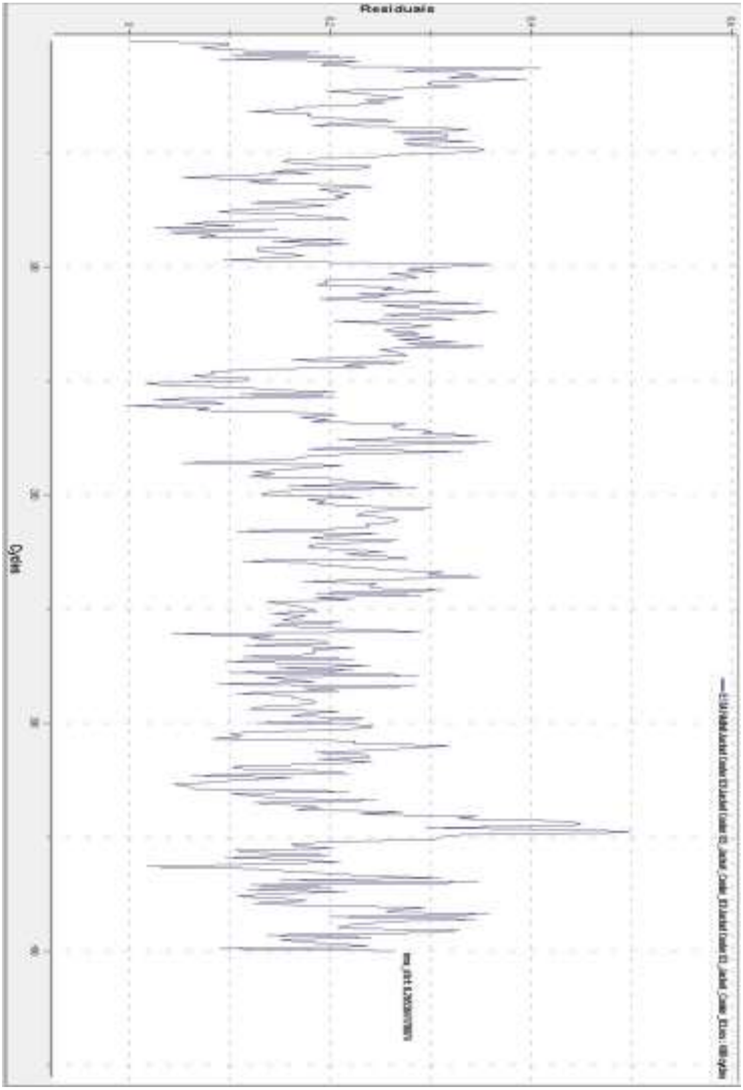
## ATTACHMENT 17

Pressure in jacket cooling system (2200 rpm)



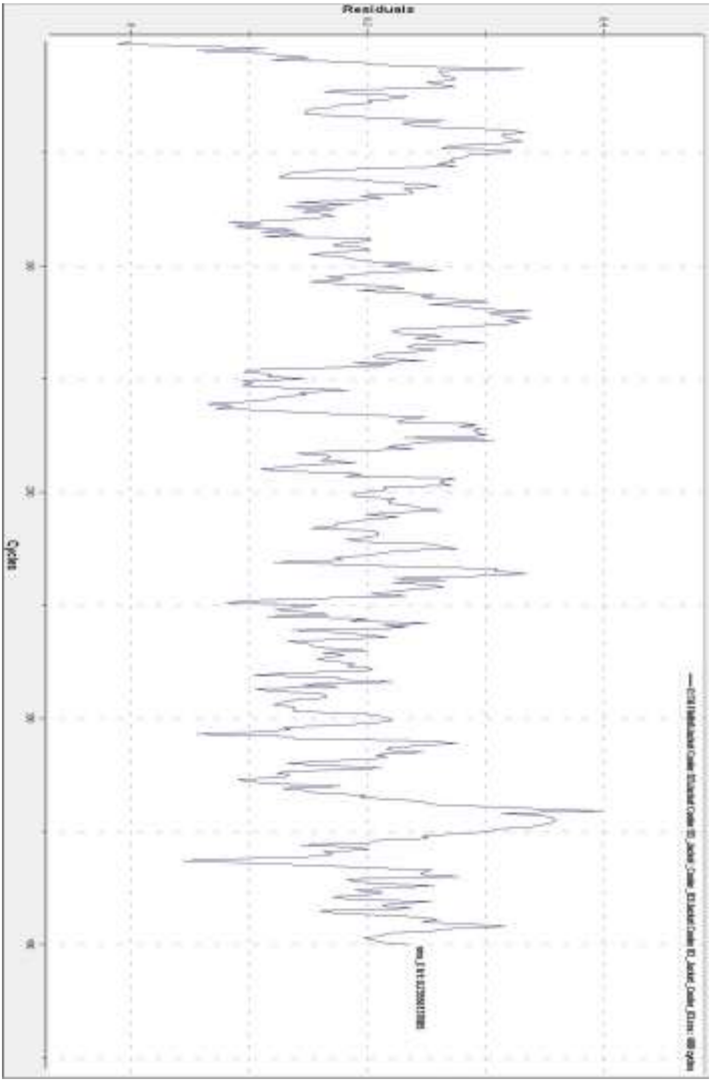
**ATTACHMENT 18**

Residuals density (2200 rpm)



## ATTACHMENT 19

Residuals energy (2200 rpm)



**ATTACHMENT 20**

HxNuCorr Objects output data with 1800 rpm

| Object Name | Rad_Experimental_int | Rad_Experimental_ext |
|---|---|---|
| Laminar Re Number Limit | 7628 | 10648 |
| Transition Re Number Limit | 7628 | 10726 |
| Laminar Exponent | 0.8110 | 0.9241 |
| Transition Exponent | 0.7419 | 0.9803 |
| Turbulent Exponent | 0.8473 | 0.2909 |
| Turbulent Coefficient | 0.0328 | 146.3830 |
| Ref. Length [m] | 0.01 | 0.01 |
| Heat Transfer Area [m^2] | 0.50 | 1.50 |
| Flow Area [m^2] | 0.01 | 0.20 |

**ATTACHMENT 21**

Heat exchanger engine block output data with 1800 rpm

| Part Name | Block |
|---|---|
| Mass Flow Rate [g/s] | 2452.4 |
| Fluid Temperature [K] | 433.8 |
| Wall Temperature [K] | 468.2 |
| Heat Transfer [kW] | 59.68 |

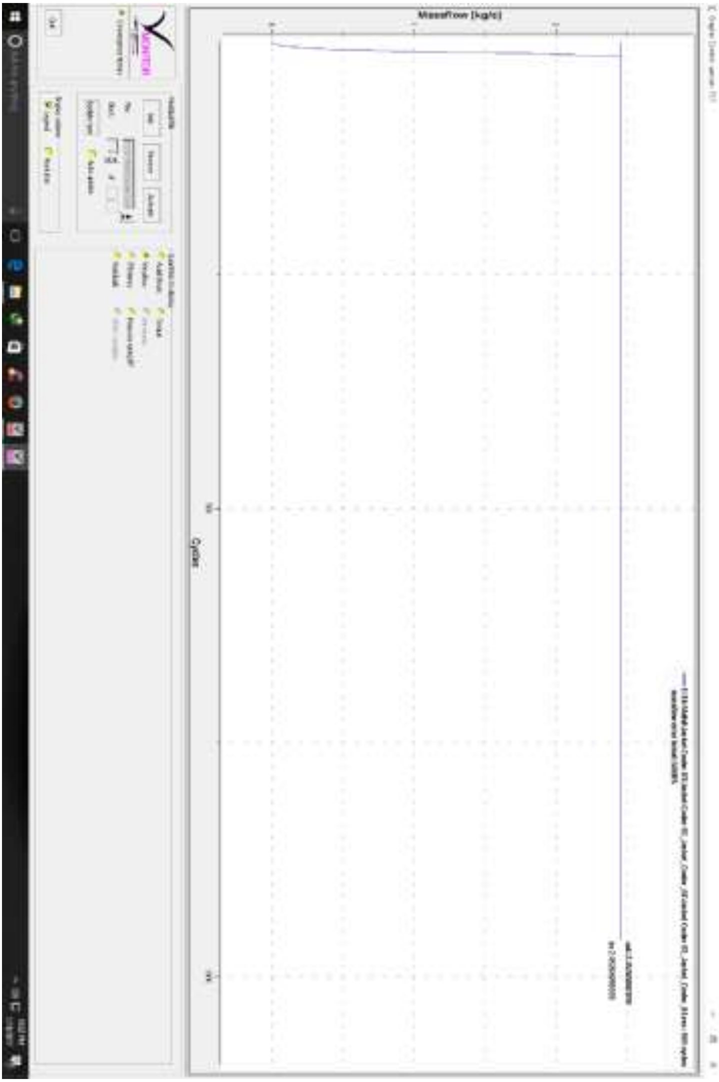
**ATTACHMENT 21**

Heat exchanger pair output data in radiator water side and radiator air side with 1800 rpm

| | MASTER | SLAVE |
|---|---|---|
| Part Name | Rad-Water-Side | Rad-Air-Side |
| Heat Transfer Object | Rad_Experimental | Rad_Experimental |
| Mass Flow Rate [g/s] | 2015.6 | 617.3 |
| Volumetric Flow Rate [liter/s] | 2.0 | 613.0 |
| Fluid Temperature [K] | 425.9 | 393.5 |
| Wall Temperature [K] | 395.9 | 395.9 |
| Pressure [bar] | 1.1 | 1.0 |
| Density [kg/m^3] | 1000.68 | 0.89 |
| Heat Transfer [kW] | -58.28 | 58.28 |
| Effectiveness [%] | 69.31 | 69.31 |
| Volume [liter] | 2.00 | 16.00 |
| Transfer Units | 1.25 | 1.25 |
| Flow Capacity Ratio | 0.08 | 0.08 |

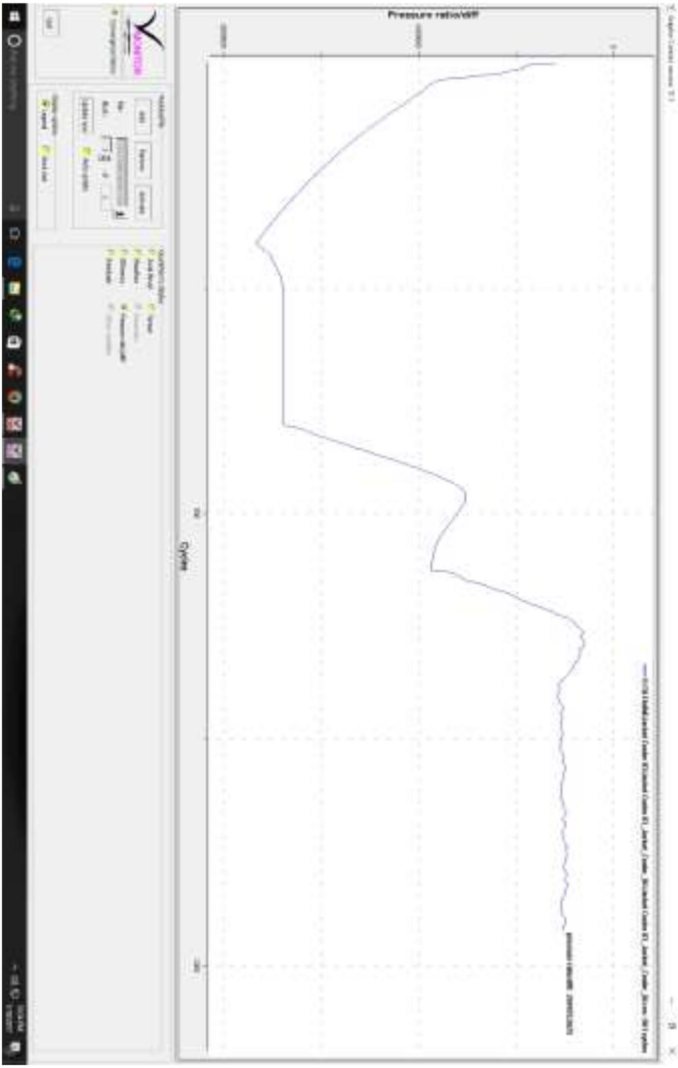
# ATTACHMENT 22

mass flow rate in jacket cooling system with 1800 rpm



# ATTACHMENT 23

Pressure ratio for jacket cooling system with 1800 rpm



**ATTACHMENT 24**

Heat exchanger pair output data in radiator water side and radiator air side with 1400 rpm

| | MASTER | SLAVE |
|---|---|---|
| Part Name | Rad-Water-Side | Rad-Air-Side |
| Heat Transfer Object | Rad_Experimental | Rad_Experimental |
| Mass Flow Rate [g/s] | 2015.8 | 425.4 |
| Volumetric Flow Rate [liter/s] | 2.0 | 443.9 |
| Fluid Temperature [K] | 461.0 | 428.8 |
| Wall Temperature [K] | 432.3 | 432.3 |
| Pressure [bar] | 1.1 | 1.0 |
| Density [kg/m^3] | 1000.68 | 0.81 |
| Heat Transfer [kW] | -55.51 | 55.50 |
| Effectiveness [%] | 75.53 | 75.53 |
| Volume [liter] | 2.00 | 16.00 |
| Transfer Units | 1.48 | 1.48 |
| Flow Capacity Ratio | 0.06 | 0.06 |

## ATTACHMENT 25

Heat exchanger engine block output data with 1400 rpm

| Part Name | Block |
|---|---|
| Mass Flow Rate [g/s] | 2452.6 |
| Fluid Temperature [K] | 468.5 |
| Wall Temperature [K] | 501.6 |
| Heat Transfer [kW] | 57.40 |

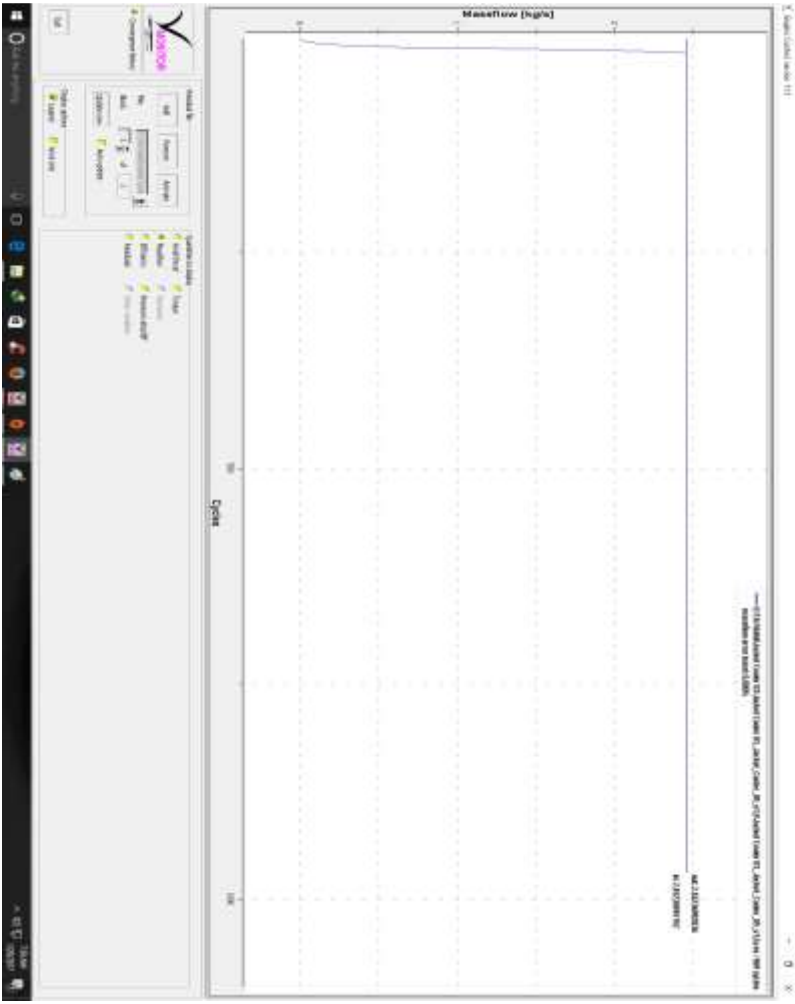
## ATTACHMENT 26

HxNuCorr Objects output data with 1400 rpm

| Object Name | Rad_Experimental_int | Rad_Experimental_ext |
|---|---|---|
| Laminar Re Number Limit | 7628 | 10648 |
| Transition Re Number Limit | 7628 | 10726 |
| Laminar Exponent | 0.8110 | 0.9241 |
| Transition Exponent | 0.7419 | 0.9803 |
| Turbulent Exponent | 0.8473 | 0.2909 |
| Turbulent Coefficient | 0.0328 | 146.3830 |
| Ref. Length [m] | 0.01 | 0.01 |
| Heat Transfer Area [m^2] | 0.50 | 1.50 |
| Flow Area [m^2] | 0.01 | 0.20 |

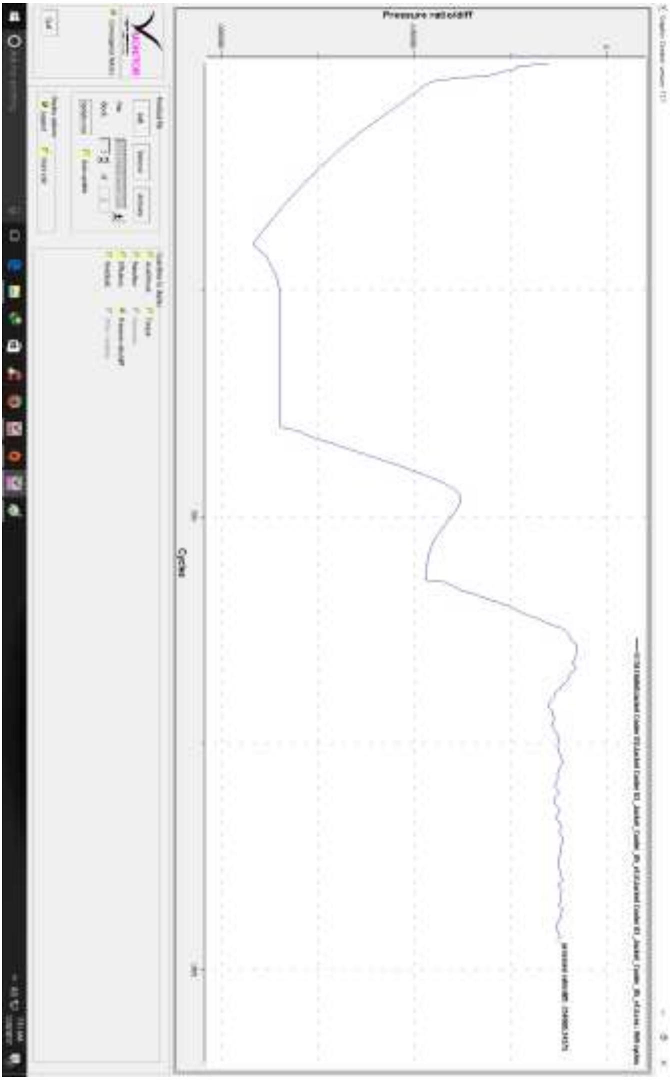
## ATTACHMENT 27

Mass flow rate of jacket cooling system with 1400 rpm



# ATTACHMENT 27

Pressure ratioa of jacket cooling system with 1400 rpm



## AUTHOR'S BIOGRAPHY

The author was born in Clausthald-Zellerfeld, Germany, june $14^{th}$, 1994, is the first child of two brothers and one sister. Author of formal education is in TK. Dharma wanita, Makassar in, SD Islam Terpadu Ar-Rahmah perumahan dosen, makassar SMP Islam Athirah Bukit baruga, Makassra in 2006, SMA Negeri 21 Makassar in 2009. Graduated from the State SMA Negeri 21 Makassar in 2012, the author continue to Double Bachelor Degree program with a major in Marine Engineering FTK - ITS through the Double Degree in 2012. Registered with the Student Registration Number 4212101023. The Department of Marine Engineering author takes a field of study Marine Power Plant (MPP). Among them are Members of Marine Engineering Student Association ITS (HIMASISKAL ITS).